ST-Realizer® II

USER MANUAL

June 1999

Ref: DOC-ST-REALIZER-II

USE IN LIFE SUPPORT DEVICES OR SYSTEMS MUST BE EXPRESSLY AUTHORIZED.

STMicroelectronics PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF STMicroelectronics. As used herein:

1. Life support devices or systems are those which (a) are intended for surgical implant into the body, or (b) support or sustain life, and whose failure to perform, when properly used in accordance with instructions for use provided with the product, can be reasonably expected to result in significant injury to the user.

2. A critical component is any component of a life support device or system whose failure to perform can reasonably be expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.

TABLE OF CONTENTS

1 INSTALLING ST-REALIZER II 1

1.1 What You Need to Install ST-Realizer II 1

1.2 Installation Procedure 1

1.3 Folders and Sub-folders 2

2 INTRODUCTION AND CONCEPTS 3

2.1 ST-Realizer Application Structures 4

2.2 Programming using Symbols 4

2.3 Inside ST-Realizer Applications 5

2.4 Symbols 6

2.4.1 State Machine Symbols 6

2.5 Schemes 7

2.5.2 The Root Scheme 7

2.5.3 Subschemes 7

2.6 Events 8

2.6.4 Execution Conditions 8

2.6.5 Event Symbols 9

2.7 How ST-Realizer Keeps Track of Time 10

2.8 Connecting your Application to the Target Device 11

2.9 Application Development Steps 11

3 TUTORIAL 13

3.1 Setting Up Your Project 13

3.1.1 Creating the Project File 13

3.1.2 Choosing a Target Microcontroller 14

3.1.3 Opening the Main Scheme 16

3.1.4 The Worksheet Toolbar 16

3.2 Designing and Drawing Schemes 17

3.2.5 The State Machine Diagram 18

3.3 Completing the Heating Control Application 26

3.3.6 What differentiates a main scheme from a subscheme? 27

3.3.7 External Input Conditions 28

3.3.8 Internal Input Conditions 32

3.3.9 External Actions 33

3.3.10 The Creation of a Subscheme for an Internal Event-Driven Action . . 35
3.3.11 Connecting Hardware Ports and Peripherals 38
3.3.12 Event Control 41
3.3.13 Summary 43
3.4 Analysing and Generating Program Code 44
3.4.14 Setting the Compile Options 44
3.4.15 Executing the Analysis and Compile 45
3.4.16 Viewing the Analyse and Compile Report 46
3.5 Simulating and Fine-Tuning Your Application 47
3.5.17 Creating the Simulation Environment File (.sef) 47
3.5.18 Connecting Probes and Adjusters 48
3.5.19 Running the Simulator 53
3.6 Now it's Up to You! 53

4 CREATING, OPENING AND SAVING PROJECTS 55

4.1 Project Files 55
4.2 Creating a New Project 55
4.3 Opening an Existing Project 56
4.4 Opening Earlier Realizer Version Projects 56
4.5 Closing a Project 57
4.6 Saving Projects 57

5 SPECIFYING THE TARGET HARDWARE DEVICE 59

5.1 ST6 or ST7 Devices 59
5.2 Choosing a Target Microcontroller 60
5.2.1 Selecting the Target Microcontroller for a New Project 60
5.2.2 Changing the Target Microcontroller 61
5.3 Hardware Configuration 62
5.3.3 Accessing Hardware Settings Dialog Boxes 62
5.3.4 General Hardware Configuration 63
5.3.5 Memory Configuration 65
5.3.6 Enabling Peripherals 66

6 CREATING, OPENING, SAVING SCHEMES 67

6.1 Schemes 67
6.2 Creating a New Scheme 67
6.2.1 Opening the Root Scheme 67
6.2.2 Creating Subschemes and other Schemes 68
6.3 Opening a Scheme 69
6.4 Saving Schemes 69

7 BUILDING SCHEMES 71

7.1 Schemes and their Components 71

7.2 Symbols 71

7.2.1 Placing and Controlling Symbols 72

7.2.2 Wiring Symbols Together and Connecting Application Inputs/Outputs 78

7.3 Working in Schemes 82

7.4 Subschemes, Execution Conditions and Events 85

7.4.3 Description of ST-Realizer Events 85

7.4.4 Execution Conditions 89

7.4.5 Event Symbols 89

7.4.6 Compatibilities Between Types of Events and Certain Symbols 91

7.4.7 Subscheme Operations 92

7.5 Table Symbols 99

8 THE MAIN SYMBOL LIBRARY 103

8.1 Input and Output Symbols 103

8.2 Sequential Symbols 105

8.3 Logic Symbols 107

8.4 Time Related Symbols 116

8.5 Mathematical Symbols 118

8.6 Counter Symbols 120

8.7 Conversion Symbols 121

8.8 Table Symbols 124

8.9 Power Management 125

8.10 Constant Symbols 126

8.11 State Machine Symbols 126

8.12 Hierarchical Sheet Symbols 128

8.13 Title Symbols 128

9 ANALYSING AND GENERATING YOUR APPLICATION 129

9.1 Overview 129

9.2 Changing the Compile Options 129

9.3 Executing the Analysis and Compile 133

9.4 What to Do if there are Errors Found during Analyse 134

9.5 Viewing and Tracing Generated Messages 135

9.5.1 Viewing the Analyse and Compile Report 135

9.6 Printing Reports 138

10 SIMULATING YOUR APPLICATION 139

10.1 Working with Simulation Environment Files 139
10.1.1 Creating a New `.sef` File 139
10.1.2 Opening an Existing `.sef` File 141
10.1.3 Saving an .SEF File 142
10.2 Setting, Adjusting and Viewing Input Values 142
10.2.4 Setting Fixed Input Values 143
10.2.5 Setting Variable Input Values 146
10.2.6 Setting Sinusoidal Input Signals 148
10.2.7 Setting Square Wave Input Signals 150
10.3 Monitoring Signals with Probes 151
10.3.8 Viewing Signal Values Numerically 152
10.3.9 Viewing Signal Values Graphically 153
10.3.10 Viewing State Machine States 156
10.4 Selecting Adjusters and Probes 157
10.5 Running the Simulator 157
10.5.11 Starting/Stopping the Simulation 158
10.5.12 Setting Run Options 158
10.6 Recording and Reusing Adjuster and Probe Values 160
10.6.13 Recording Adjuster and Probe Values 161
10.6.14 Reusing Adjuster Values 162

11 CREATING YOUR OWN SYMBOL 165

11.1 Overview 165
11.2 Running the ST Symbol Editor 165
11.3 Defining a New Subscheme Symbol 166
11.3.1 Adding Your New Subscheme Symbol to a Library 169
11.4 Defining a New User-Defined Symbol 171
11.4.1 Defining the New Symbol 172
11.4.2 Editing the New Symbol 176
11.4.3 Adding Pins to Your Symbol 181
11.4.4 Assigning Attributes to Your Symbol 182
11.4.5 Modifying Existing Attributes 185
11.4.6 Creating the Macro Header 187
11.4.7 Creating the New User-Defined Symbol Macro 187
11.4.8 Writing the Assembly Macro 188
11.4.9 Adding New User-Defined Symbols to a Library 191

12 CUSTOMIZING ST-REALIZER 193

12.1 Automatically Saving Your Work and Setting Screen Preference. 194

12.2 Attribute Display Preferences 195

12.3 Worksheet Layout Preferences 196

12.4 Printing Options 197

12.5 Symbol Layout Preferences 198

12.6 Customizing Toolbars 199

12.6.1 Adding and Deleting Toolbar Buttons 200

12.6.2 Placing Separators Between Toolbar Buttons 200

12.6.3 Changing the Order of Toolbar Buttons 200

12.6.4 Restoring the Default Toolbar 200

12.7 Wire Drawing Options 201

Appendix A:Variables and Attributes 203

A1 Variable Types and Rules 203

A1.1 Type Inheritance 204

A1.2 Type Overruling 205

A2 Attribute Types 205

A2.1 Pin Attributes. 206

A2.2 Symbol Attributes 207

Appendix B:Sample Applications 211

B1 Coded Lock Application 211

B1.1 Application Overview 211

B1.2 Functional Description 212

B1.3 Sequencing Control 212

B1.4 Secret Code Storage in the EEPROM 214

B1.5 Access Code Entry and Recognition 214

B2 Analog Multiple Key Decoder 215

B2.1 Application Overview 215

B2.2 The Keyboard 216

B2.3 Software Generation 217

B2.4 Possible Improvements. 218

B3 Clock Design 220

B3.1 Application Overview 220

B3.2 Current Time Counting 220

B3.3 Current Time Setup. 221

B3.4 Alarm Time Setup 221

B3.5 Alarm triggering. 221

B3.6 Timebase 221

B3.7 Current Time Counting 222
B3.8 Current Time Setup. 223
B3.9 Alarm Time Setup 224
B4 Fast Counter Application 225
B4.1 The Application 225
B4.2 Fast Counter Report File. 228
B4.3 Generated Code 230
INDEX 237

1 INSTALLING ST-REALIZER II

1.1 What You Need to Install ST-Realizer II

You must install ST-Realizer II on a PC that meets the following requirements:

Table 1 PC Hardware Requirements

Minimum requirements	For Optimum Performance
Processor: Intel®80486 RAM: 8 Mb Disk memory: 14 Mb Monitor: Grey scale VGA Mouse	Processor: Intel® Pentium-100 MHz RAM: 16 Mb Disk memory: 16 Mb Monitor: Super-VGA, 17" Mouse

ST-Realizer II runs under Microsoft® Windows® 95, 98, or NT®.

1.2 Installation Procedure

1 Boot your PC under Windows.

2 Put the ST-Realizer CD-ROM in your CD-ROM drive.

The CD-ROM's autorun function will open the Setup program automatically.

3 Follow the instructions that appear in the pop-up windows.

The Installation program will ask you to specify the folder into which you wish to install ST-Realizer. The folder you choose will be the ***root folder***. Either accept the default or enter a new installation folder.

Installation is now complete.

To launch ST-Realizer, click **Start→Programs→ST-Realizer II→Realizer**.

1.3 Folders and Sub-folders

The installation process creates the following folders and sub-folders:

<root_folder> which contains the system executable files and DLLs. (The root folder is **`C:/Program Files/ST-Realizer/`** by default.)

<root_folder>\Examples which contains examples of ST-Realizer projects.

<root_folder>\Help, which contains the help files, a readme file and PDF documents about ST-Realizer.

<root_folder>\Lib, which contains the main symbol library and a number of other libraries.

<root_folder>\TargetHW, which contains the definition files for the ST6 and ST7 microcontrollers supported by ST-Realizer II.

2 INTRODUCTION AND CONCEPTS

The founding idea behind ST-Realizer was to create an accessible and user-friendly software package, allowing people at various levels of programming expertise to efficiently design embedded applications for ST6 and ST7 microcontrollers.

ST-Realizer is an application programming package that allows you to create applications ready to be loaded into ST6 and ST7 microcontrollers without having any knowledge of assembler code. To do this, you use symbols that represent programming functions to create flow diagrams that perform your application functions. While the user is assumed to have a good understanding of the microcontroller for which he or she wishes to create an application, care has been taken to create a sufficiently broad spectrum of symbols to cover all of your application design needs. And should you require a symbol not included in ST-Realizer's main library, you can design your own using the Symbol Editor function.

All ST-Realizer applications are destined for one of the ST6 or ST7 family of microcontrollers. The scope of the application is necessarily limited by the resources available on the **target device**—the microcontroller for which the application has been designed.

It is therefore imperative that you fully understand the specifications of the target microcontroller before you begin to design your application. Datasheets for those ST6 and ST7 microcontrollers supported by ST-Realizer are supplied on the ST-Realizer CD-ROM. In addition, datasheets for ST microcontrollers can be easily obtained from the STMicroelectronics microcontroller web site:

http://st7.st.com

The remainder of this chapter will describe the basic concepts behind using ST-Realizer, to help you generate your embedded application programs.

ST-Realizer was developed by ACTUM Solutions expressly for STMicroelectronics, for use in developing embedded applications for ST6 and ST7 microcontrollers. In addition to ST-Realizer, ACTUM Solutions provides a variety of other software products, some of which can be used as a complement to ST-Realizer to further refine your application. For more information, please refer to the ACTUM Solutions web site.

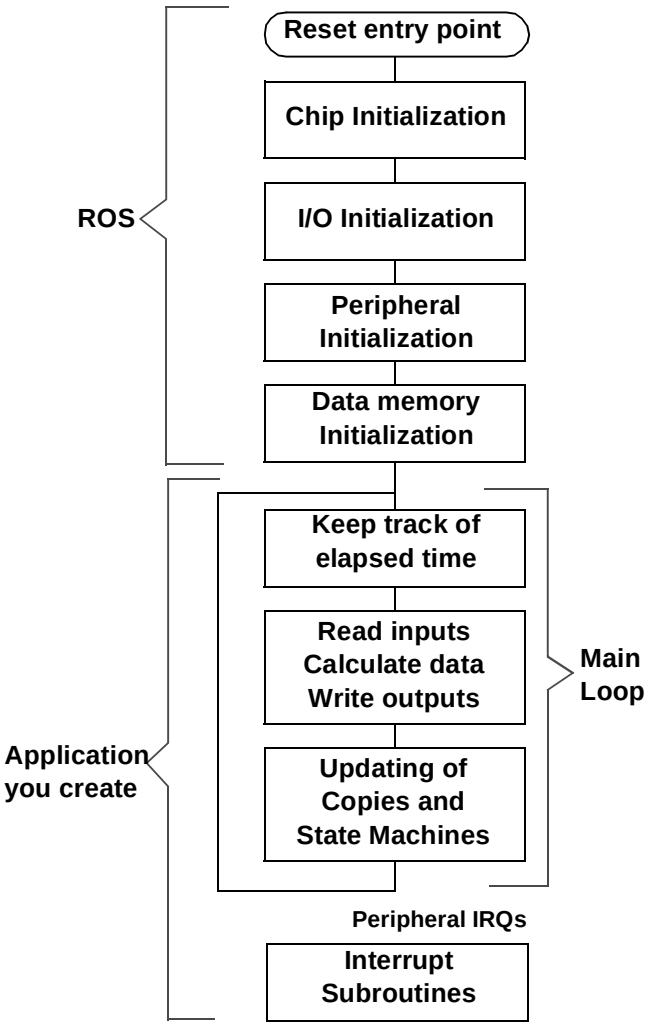
http://www.actum.com/

2.1 ST-Realizer Application Structures

Perhaps the best place to start describing ST-Realizer is at the final product—the generated assembler application that ST-Realizer will produce for you. It is important to understand how the final generated code is structured before you start designing your application, so that you are aware of how best to optimize available resources, such as memory.

There are two main parts to each ST-Realizer assembler application. The first part is a series of initialization macros that are embedding automatically and that make up the Realizer Operating System (ROS). The ROS sequentially initializes the microcontroller, it's I/O's, peripherals and memory, in much the same way that your PC's BIOS initializes the PC hardware as soon as you switch the power on. The second part of the code is the part that you create using ST-Realizer—the application program.

The figure at right shows a flow chart of the overall structure of the generated assembler code that ST-Realizer produces.



2.2 Programming using Symbols

With ST-Realizer, you create applications by placing and connecting ***symbols*** in a ***scheme***. Each symbol is, in fact, a graphical representation of an assembler macro, usually including attributes which you can modify to your specific needs.

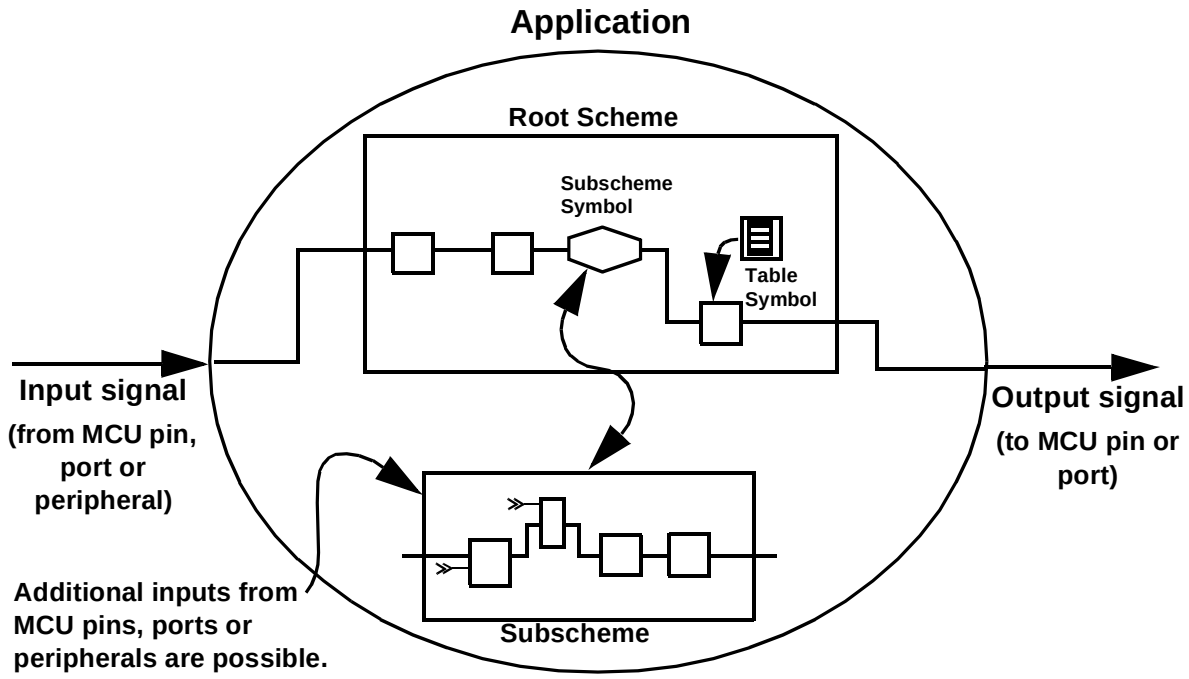
The symbols included in the ST-Realizer main library represent a variety of coded entities such as: mathematical, logical, conversion and power management functions, constants, tables, subschemes/hierarchical sheets, states, input devices, output devices and sequential, counted or time-related events.

The symbols are made such that you need never write a single line of assembler code to produce your application—all of the attribute modifications you may need to perform are accessible through dialog boxes.

2.3 Inside ST-Realizer Applications

An application is built around an ST6 or ST7 microcontroller unit (MCU). The input signal(s) enter the application from one (or more) of the microcontroller's pins, ports or peripherals. The application treats the input signal(s) as you require, and the result is output to one of the microcontroller's pins or ports.

The figure below shows a generalized view of an application. The input signal(s) enter the application via one of the MCU's pins, ports or peripherals, and is taken up by the **root (or main) scheme**. The root scheme is the core of the application—the main, sequential loop. If the application requires any interrupts, you must create a subscheme. (Interrupts cannot occur in the root scheme.) If you simply wish to section off a very complex part of the application for aesthetic reasons, you may also create subschemes to contain parts of the main loop. Subschemes are represented in the root scheme by ***subscheme symbols***, of which more will be said a little later.



2.4 Symbols

Using ST-Realizer, you design your application by placing symbols and wiring them together in schemes.

Each symbol may represent:

- An **operation**, such as converting a physical analog value to a binary value,
- A **piece of information** related to the **behavior** of the application, such as a state transition,
- A **system state**, or **condition**,
- An **action** reflecting a change in the **system state** or caused by an **event** such as the occurrence of a timer interrupt.

Each symbol is associated with an ST6 or ST7 assembler code macro. The wires represent the flow of data, and are linked to variables and constants. You can modify certain attributes of symbols and wires, allowing you to customize them for your specific application. For example, by attaching an attribute of type UINT (unsigned integer) to a wire, you define its value capacity to that of an unsigned integer (0 to 65536). For more details on attributes see Appendix A: “Variables and Attributes” on page 203.

2.4.1 State Machine Symbols

Within your root scheme, you may create a **state machine**, which logically guides the program between different functional states of your application. Say, for example, you had an application which performs the following functions:

- Turning a motor on.
- Setting the motor speed.
- Turning the motor off.

In a state machine, you would define a state, using a **state symbol**, for each functional step of the application above and, in addition, a state which defines the starting point of the application—the initial state.

The sequence of state symbols would therefore look like:

- Motor OFF (Initial State).
- Motor ON.
- Setting Speed.
- Motor OFF.

The transitions between each of these states are controlled by **conditions**. **Condition symbols** act as switches. When a condition is met, the condition symbol is triggered, and the program can progress to the next state.

The tutorial included in this manual (see Chapter 3 “Tutorial” on page 13) provides an very good example of how a state machine and state symbols can be used in creating an application.

2.5 Schemes

When using ST-Realizer, you design your application in **schemes**. A scheme is like a plan on which you place symbols and draw wires. Each application consists of a set of schemes, including one root scheme and any number of subschemes. Section 7 on page 71 explains how to build and modify schemes.

2.5.2 The Root Scheme

The root scheme is the starting point of your application program, and corresponds to the reset vector of the program.

The root scheme is where you create the main loop of your application. All of the large scale, sequential functions should be kept here. However, if there are any particularly complicated or cumbersome actions in your main loop, you may wish to put them into a subscheme to save space in the root scheme and to make the application easier to follow visually.

2.5.3 Subschemes

Applications can include any number of **subschemes** which contain further symbols and wires but are displayed in the root scheme as a single symbol.

There are three reasons to create a subscheme:

- To include complex portions of the main loop, thus saving space in the root scheme and making it easier to reuse processes. In this case, the subscheme is executed as if it were a part of the main loop (root scheme).
- To include parts of the application that are event-driven. (Events can never be placed in the root scheme.) Subschemes can be assigned either a single **execution condition**, which will apply to the entire subscheme, or alternatively, can include any number of **event symbols**. More will be said about execution conditions and events shortly.
- To save functional parts of your application (analogous to subroutines) that you may wish to reuse in other applications. Subschemes are saved in their own files (**.sch** files) and can be easily copied to other ST-Realizer projects and reused. You may also save customized subschemes symbols to a library, to be accessible by all projects. (Subscheme symbols are described below).

Designing a subscheme is no different than designing an ordinary scheme, with one exception: a subscheme has connections to its root scheme via a ***subscheme symbol***. The subscheme symbols are named **sssp_q**, where **p** indicates the number of inputs you need for your symbol and **q** the number of outputs. For example, **sss2_1** is a subscheme with two inputs and one output.

When you want to use a subscheme, you must therefore first think about its connections: what inputs does the subscheme need to deliver its output? Once you know this, you can choose the correct subscheme symbol from the main library. However, subschemes, like the root scheme, can be modified at any time. Section 7.4 on page 85 describes how to create and modify subschemes.

2.6 Events

Events are conditional triggers, similar to If..Else statements, that can be applied either to an entire subscheme, or simply to a sequence of code. Like an If..Else statement, events are always triggered by an input of some sort. The input may be:

- An interrupt, such as a timed or hardware interrupt.
- An input value change.

When an event is applied to an entire subscheme, it is called an ***execution condition***—because it defines the conditions by which the subscheme will be executed.

However, events can also be made to apply to just a sequence of symbols within a subscheme, using ***event symbols***. These symbols act as switches—if the condition that they represent is met, the code that follows them can be performed.

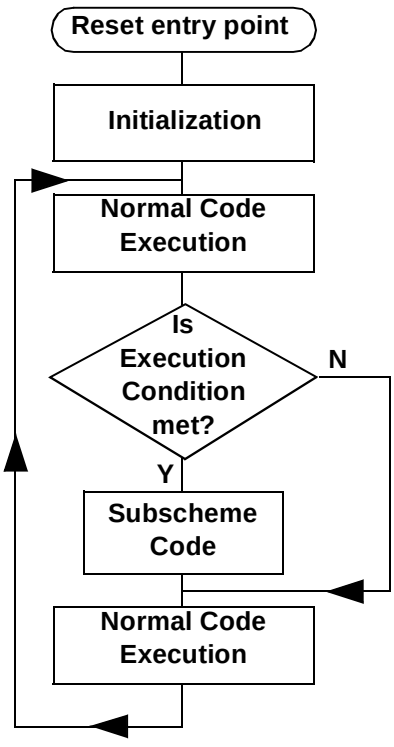
There are many types of events, some hardware independent, and others that are hardware dependent. The full range of events available is detailed in Section 7.4 on page 85.

2.6.4 Execution Conditions

An execution condition can be applied to a subscheme, such that the subscheme is only executed when that execution condition is met—such as a timed interrupt, or upon a subscheme input change. Only one execution condition can be applied to any given subscheme and when this execution condition is fulfilled, all of the code within the subscheme is executed.

Subschemes with execution conditions are chiefly used to contain reasonably complex subroutine functions that are conditionally performed in addition to the code in the main programming loop.

The diagram at right shows a schematic example of how a subscheme with execution conditions is used in an application.



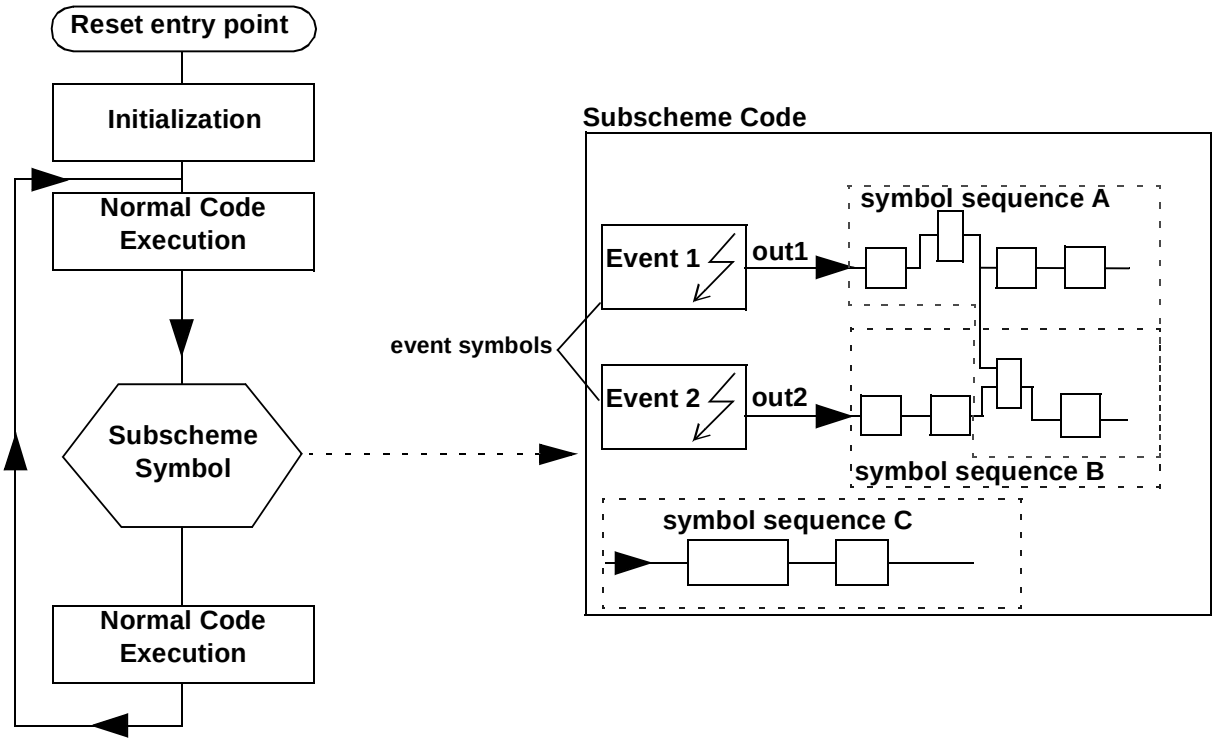
2.6.5 Event Symbols

Event symbols can be included in subschemes to determine when, and which portions of, the code is executed. Event symbols are always placed in subschemes, because events act as interrupts, and interrupts may never be placed in the root scheme.

In a certain manner, event symbols act as switches to control when (i.e. under which conditions) subsequent code is executed.

Event symbols are most usefully used when you wish to include several events that may control the similar portions of code. However, certain rules apply when placing more than one event symbol in a single subscheme (refer to Section 7.4.5 on page 89).

The diagram below shows a schematic example of how event symbols can be used to control how a subscheme is executed.



In the above example, the subscheme code will be executed as follows:

- If Event 1 is triggered (by the event's condition being fulfilled):
  - Symbol sequence A will be performed with out1 = 1.
  - Symbol sequence B will be performed with out2 = 0.
  - Symbol sequence C will be performed unconditionally—there are no event symbols connected to this code sequence.
- If Event 2 is triggered:
  - Symbol sequence A will be performed with out1 = 0.

- Symbol sequence B will be performed with out2 = 1.
- Symbol sequence C will be performed unconditionally.

- If neither Event 1 nor Event 2 is triggered, **no part** of the subscheme code will be performed.

Note: Even though symbol sequence C is not directly connected to a event symbol, by virtue of it being in a subscheme with that contains events, it will not be performed unless one of the events is triggered. The golden rule is that you cannot mix events with root scheme symbols (meaning those symbols that are performed as part of the main loop or normal code). When you place symbols in a subscheme which contains one or more events (either in the form of event symbols or an execution condition), those symbols cannot be considered as part of the root scheme.

2.7 How ST-Realizer Keeps Track of Time

ST-Realizer II differs from its predecessors because the final code that it produces will **only** contain timer initialization code **if** there are time-related symbols or events in the application.

However, if your application includes either time-related symbols or events, ST-Realizer will generate something called a **base clock timer tick** in the following manner:

- Every ST6 and ST7 microcontroller has a timer, called Timer 1 (ST6) or Timer A (ST7), which (if there are either time-related symbols or events in the application) is used as the base clock to measure out units of time called “timer ticks”. You can choose to set the value of the timer tick—this is described on page 131.
- All time-related symbols and events are based on timer ticks. This means that one timer tick is the smallest increment of time that can be distinguished.
- Timer ticks may be used to control the processing time of a main loop cycle. The time required to perform one main loop cycle is called the ***Processing Cycle Time***. By default, the processing cycle time is variable. However, you can choose to ***fix*** the processing cycle at a specific number of timer ticks—how to do this is described on page 131.

For example, by default the base clock timer tick is set to 0.01 s (10 milliseconds). This means that every 10 milliseconds the hardware timer (Timer 1 or Timer A) sends an interrupt to the program which increments a tick variable. Time-related symbols and events use this tick variable (either directly or indirectly[(1)]) to evaluate whether their conditions have been satisfied, and whether their actions should be executed or not.

1 How different types of time-related events use the value of the tick to evaluate their conditions is detailed in Section 7.4.3 on page 85. Time-related symbols are described in detail in Section 8.4 on page 116.

The concept of the base clock timer tick is an important one, because it appears every time we require a time-related symbol or event. In our tutorial example, we demonstrate how to use both time-related symbols and timed events. We strongly urge you to take the time to complete the tutorial—it is a very efficient way to get up to speed in ST-Realizer and the time you spend doing the tutorial will be saved later by having increased your productivity!

2.8 Connecting your Application to the Target Device

All signal inputs to the application are supplied by one or more of the microcontroller's input pins, ports or peripheral control registers. Similarly, the application's final output must also be sent to an output pin or a port. In general, each ST6 and ST7 microcontroller has a variety of digital and analog input/output pins, as well as ports for serial or parallel data. The number of pins and ports, of course, depends on the microcontroller in question.

However, peripheral support can vary largely, depending on the microcontroller to be used. Any application design, must obviously bear in mind the resources available on the microcontroller.

To link inputs and outputs between the application and the microcontroller, you must connect pins, ports or peripheral control registers to input or output symbols in the application's schemes.

Note that peripherals must be **enabled** before they can be used by the application. Once enabled, each peripheral used must usually be configured to meet the hardware requirements of your application—hardware setting dialog boxes are designed for this purpose. These hardware settings are used by the ROS to initialize the microcontroller properly.

2.9 Application Development Steps

Once you have designed your application using ***ST-Realizer***, you analyse and compile it using ***ST-Analyser***.

ST-Analyser performs the following tasks:

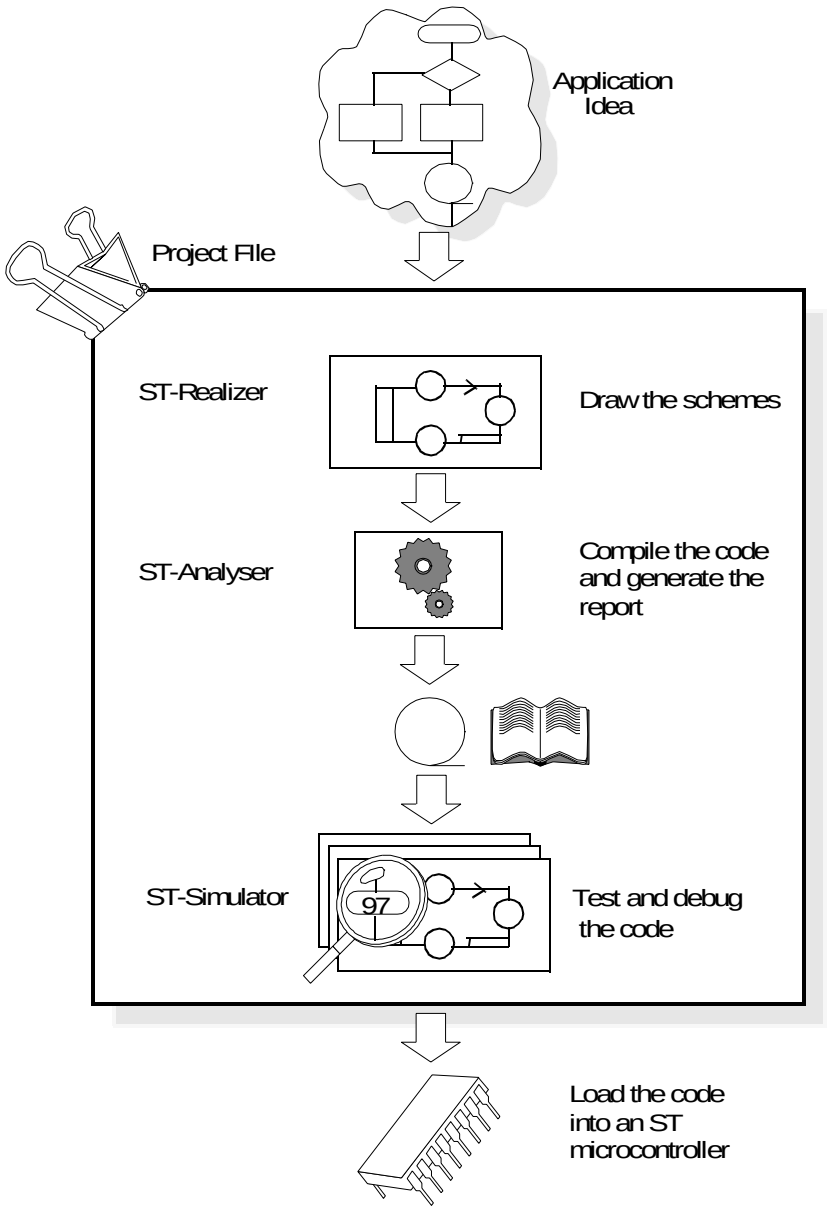
- Analyses your scheme by creating the netlist, creating cross references, analysing and generating final code. Providing no fatal errors are encountered, ST-Analyser generates a non-compiled ST6 or ST7 macro-assembler language (**`.asm`**) file from the scheme.
- Generates the compiled binary ST6 or ST7 executable file. Depending on whether or not you included the ROS (see Section 2.3 on page 5 and Section 5.3.4 on page 63), a file with extension ***.hex** or ***.obj** respectively is generated for ST6, or with extension ***.s19** or ***.obj** for ST7. A ***.hex** (or ***.s19**) file can be directly loaded into an ST MCU while you must link a ***.obj** file with another program.

When the analysing process has been successfully completed, a report file is generated. This report file gives information about the designation of I/O pins, a list of the variables used by type and the memory space required by the application.

Once you have compiled your application, you can use ***ST-Simulator*** to simulate its behavior, generate and view input signals, monitor signals that are generated by your application, and fine-tune it if necessary. You design simulation environments in the same way you design schemes, except that the design is held in what are called simulation environment files.

To provide you with greater flexibility, you can create or edit your own symbols using ***ST-Symbol Editor***. You create a symbol by drawing its shape, placing pins that represent the variables that are input to and output from the process you are defining, then linking it to the macro it represents.

All the files and definitions that pertain to an application are stored in **project files**. The following diagram shows the ST-Realizer application development process.



3 TUTORIAL

The following tutorial is designed to help you fully understand both the *principles* behind creating applications using ST-Realizer and *how to create* applications using ST-Realizer.

In this tutorial, you'll learn how to create an ST microcontroller application.

The application you will create manages a ***heating control system***. The ambient temperature is periodically measured, filtered and compared with a preset value. When the measured temperature is lower than the control value, the heating system is started while the pump speed is adjusted proportionally to the temperature differential. A pilot LED lights up to show that the heating is on. When the measured temperature exceeds the control value, the heating system is stopped.

This tutorial application runs in stand-alone mode using an ST72212G2 microcontroller. However, note that the same application could be applied equally to an ST6 microcontroller, such as the ST6265, with little modification. The goal of this tutorial is to demonstrate, in a general manner, how to use ST-Realizer, rather than to supply sample applications for a given microcontroller.

3.1 Setting Up Your Project

In this part of the tutorial you will:

- Create a new project file for the heating control application.
- Define the ST microcontroller onto which the application will be loaded.
- Learn how to open the scheme in which you'll draw the application.

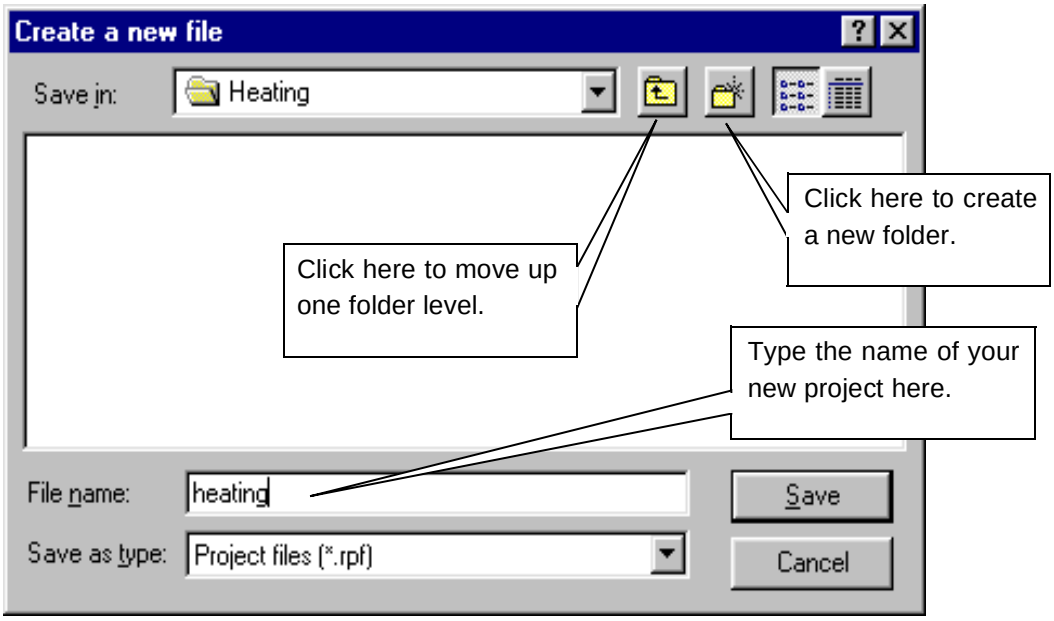
3.1.1 Creating the Project File

Each application you design is stored in a **project**. When you create a new project, the first step is to specify the folder in which you wish to place it. It is recommended that you create a separate folder for each new project.

ST-Realizer will creates a ***project.rpf*** file, that contains project-specific path settings, the project's scheme names, target hardware information and compiler settings.

Once you've defined your project, you'll be able to open a new scheme and start designing your application.

1 Click *Project→New* in the cascading menu. The *Create a New File* dialog box opens:



Create a new folder for your tutorial project called "**Heating**" by clicking on the new folder icon shown above.

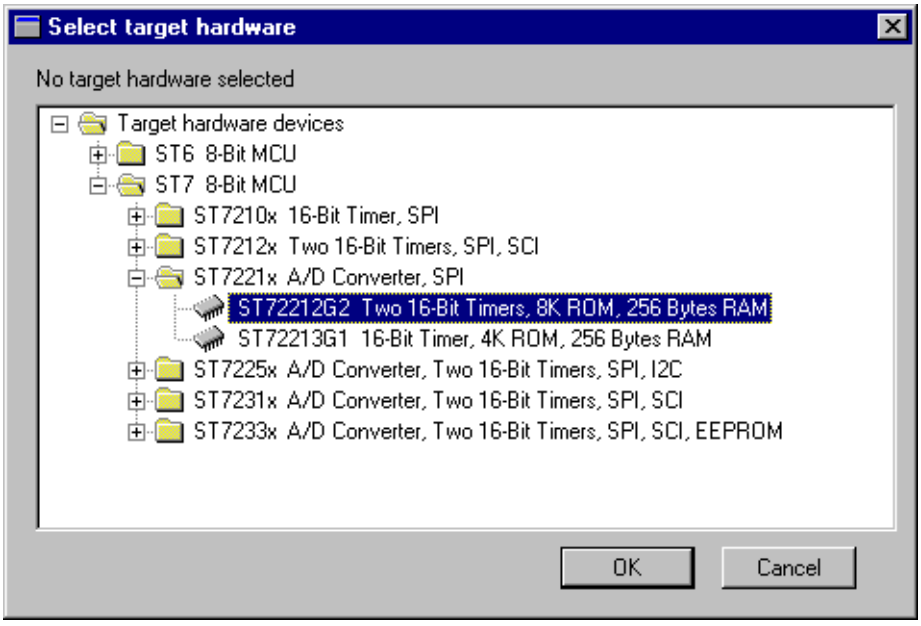
2 Once you have created your project folder, type the name of the project ("`heating`") in the *File name* field. ST-Realizer will add the `.rpf` extension automatically. Click *Save*.

You have just created the **`heating.rpf`** file in which the main settings related to your project will be recorded at the end of the ST-Realizer session. They will be used the next time you open the project.

3.1.2 Choosing a Target Microcontroller

The next step is to choose the ST microcontroller on which the application will be loaded. After you create a new project, a window will open prompting you to select the target hardware. This application is going to be loaded on an ST72212G2:

1 **Click the *Target Hardware* folder in the Select Target Hardware window. A browsable list of target hardware devices will appear in the window shown below.**

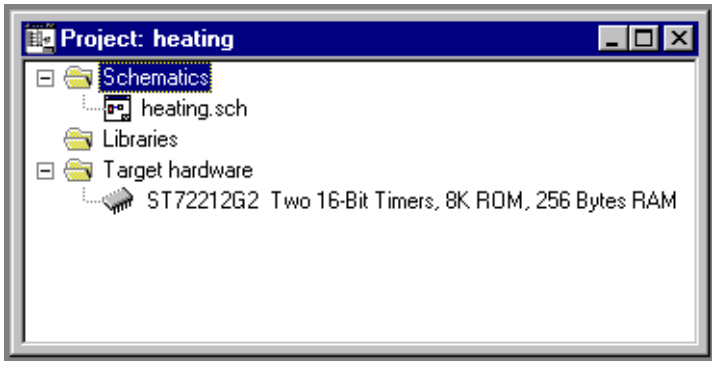

Select target hardware
No target hardware selected
Target hardware devices
ST6 8-Bit MCU
ST7 8-Bit MCU
ST7210x 16-Bit Timer, SPI
ST7212x Two 16-Bit Timers, SPI, SCI
ST7221x A/D Converter, SPI
ST72212G2 Two 16-Bit Timers, 8K ROM, 256 Bytes RAM
ST72213G1 16-Bit Timer, 4K ROM, 256 Bytes RAM
ST7225x A/D Converter, Two 16-Bit Timers, SPI, I2C
ST7231x A/D Converter, Two 16-Bit Timers, SPI, SCI
ST7233x A/D Converter, Two 16-Bit Timers, SPI, SCI, EEPROM
OK
Cancel

2 **Find the desired ST microcontroller (ST72212G2) by clicking the device icons in the list until the name of the chosen device is displayed. Select the ST microcontroller ST72212G2 by clicking once on it.**

3 **Click OK to confirm.**

You may also double-click the line showing the device.

4 **Once you have selected your target hardware, the Project window will open, as shown below:**



This window will list all of the components of your project—schemes, libraries and target hardware. Notice that ST-Realizer has already created an application scheme file called heating.sch by default. Next, we'll learn how to design the application scheme.

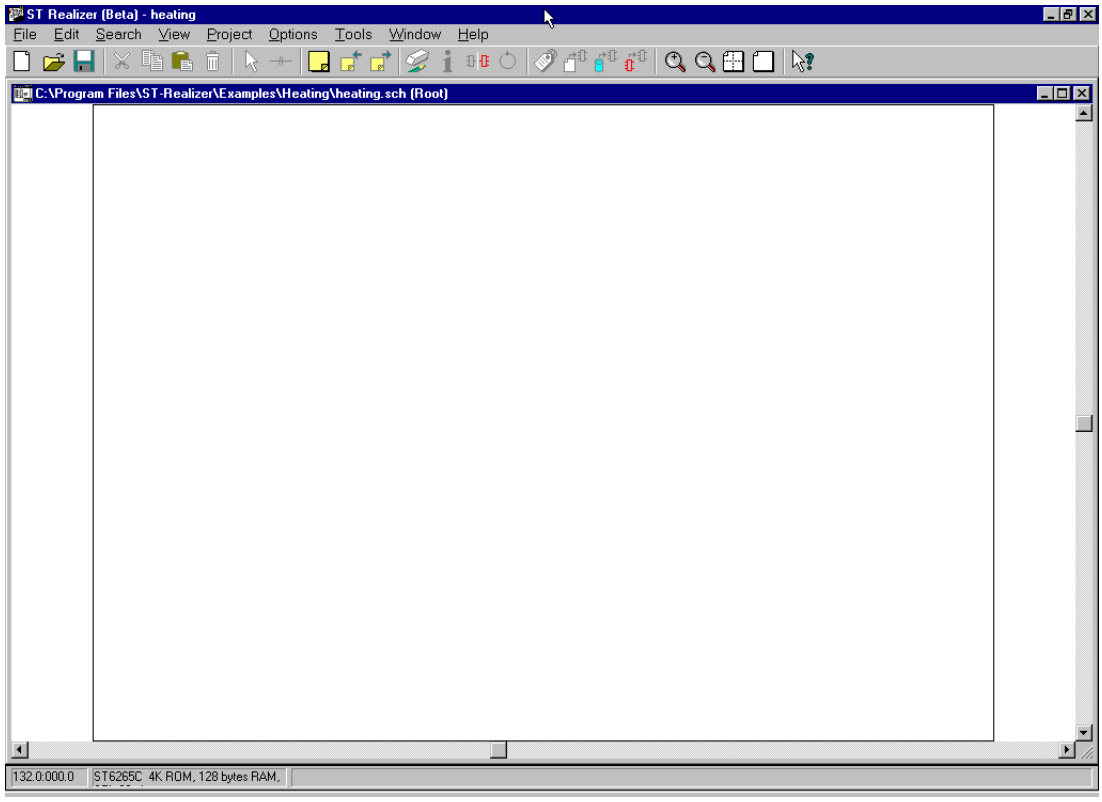
3.1.3 Opening the Main Scheme

Now that you have created your project and specified the target microcontroller, you must draw the ***main (or "root")* scheme.** This is the sheet on which you design the main part of your application. The main scheme file has already been created by ST-Realizer, and is called **`heating.sch`** by default.

To open the main scheme:

- **In the Project window (shown on page 15), double-click `heating.sch`, located under the *Schematics* folder.**

  A new blank worksheet opens where you can draw the main (root) scheme of the application:


ST Realizer (Beta) - heating
File Edit Search View Project Options Tools Window Help
C:\Program Files\ST-Realizer\Examples\Heating\heating.sch (Root)
132.0:000.0
ST6265C 4K ROM, 128 bytes RAM,

3.1.4 The Worksheet Toolbar

At the top of the scheme worksheet, you will see a toolbar with a number of icons. How to use these icons will be described as we work through the tutorial, but a quick summary of their names and uses is given here.

Note: The step-by-step instructions in this tutorial describe how to operate ST-Realizer using the default toolbar setup shown above. If, while you are doing this tutorial, you cannot find a button that is shown in an instruction, refer to "Customizing Toolbars" on page 199, to find out which menu commands correspond to that button.

3.2 Designing and Drawing Schemes

For the heating control application, the microcontroller will require the following inputs and outputs:

- 2 analog inputs:
  - The actual temperature
  - The control temperature
- 1 digital output to one pilot LED to indicate the heating system status,
- 1 digital output to control the speed of the pump.

In this part of the tutorial, you are going to learn how to create the above application by drawing graphical schemes. In particular, you will learn:

- How to place symbols.
- How to edit symbols.
- What the principal symbols do, and when to use them.
- How to connect the pins.
- How to handle events.

The schemes associated with the application include symbols that describe:

- A *State Machine* diagram.
- *Conditions*, that is conditions that are transmitted to the application from external sources such as temperature records, or that are internally determined, for example when the

system switches to a specific state.

- *Actions*, that is actions that are output from the application, such as putting on the pilot LED, or actions that are internally determined, for example those resulting from a timer interrupt.

The following section will explain how to draw the heating application using symbols fulfilling the above roles.

3.2.5 The State Machine Diagram

Each time a condition changes, as a result of the signals received from the microcontroller input pins, the state machine selects the appropriate state. The state defines the signals that are sent to the microcontroller output pins.

The overall management of the heating control system is carried out by a simple state machine with four states: *Init, HeatingOFF*, *HeatingON*, and *Pump Enabled*.

SetupTime, Start, SetPump and *Stop* conditions trigger state transitions.

A specific action is associated with each state.

The following table summarizes the relationship between events(1) (or inputs), conditions and actions.

Event(1) or Input Signal		**State Transition**		**Action**
Internal	External	Condition	New State	
Setup Time has elapsed		SetupTime	HeatingOFF	Pilot LED OFF
	Measured Temp. is lower than Control Temp.	Start	HeatingON	Pilot LED ON
	Measured Temp. exceeds Control Temp.	Stop	HeatingOFF	Pilot LED OFF
System switches to HeatingON state		SetPump	PumpEnabled	Pump speed control active
Event = Timer Interrupt		No state transition		Input acquisition active

1 For more information about Events, please refer to Section 7.4 on page 85.

When you define the state machine, the first symbol you must place is the **initial state**.

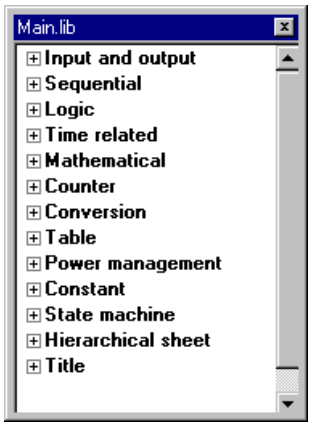
For the heating control application, the initial state is *Init*. This is a holding state that allows time for the application to measure and filter the actual temperature, before deciding whether to turn on the heat pump.

3.2.5.1 Placing the Initial State

To place the initial state, Init:

1 Click .

2 The main library dialog box will open, as shown at right:



All of the symbol names are categorized by function. A complete listing of symbols (organized by functional category and name) can be found in Chapter 8 on page 103.

The initial state symbol is called “stateinit”, located under the “State machine” category.

Find “stateinit” and select it by:

- **double-clicking on it,**

or,

- **clicking once on it and, without moving the cursor from the library dialog box, right-click the mouse and select *Place*.**

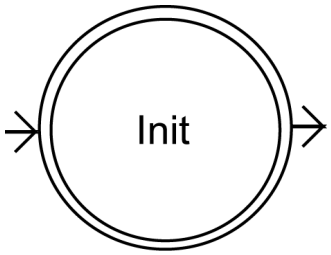
3 A square box will now appear next to the cursor, indicating the size and position of the “stateinit” symbol. Move the cursor to where you want to place the symbol, then click once. It is recommended that you place the symbol towards the top and left of your scheme.

Note: If you wish to move the symbol after having placed it, just click once next to the symbol so that a red rectangle appears around the entire symbol, indicating that it has been selected. Then simply drag and drop it where you wish.

4 A dialog box will open, prompting you to edit the value of the state. In doing so, you are effectively naming the initial state variable. Type “`Init`” in the field, then click OK.

CAUTION: With ST-Realizer, object naming (such as when you assigned the name “Init” to the initial state symbol above) is case-sensitive. In addition, spaces are interpreted as characters. Ensure that all object names are used consistently, otherwise, errors will result when you compile the application.

You've just placed your first symbol—the initial state, Init. It should look like this:



If it doesn't, select the symbol by clicking next to it, then delete it by pressing the Delete button on your keyboard, and redo steps 2 to 4 above.

If you find that the symbol is too small, and you want to zoom in on it, click , then select the area around your symbol. If you make a mistake, and can no longer see your symbol, click to see the whole scheme, then reuse to zoom in on your symbol.

Now you are ready to place the first condition—the *SetupTime* condition—that is activated for the first time when the start button is pressed. Active conditions are signalled by sending the value 1—a signal value equal to 0 indicates that the condition is not active.

3.2.5.2 Placing a Condition

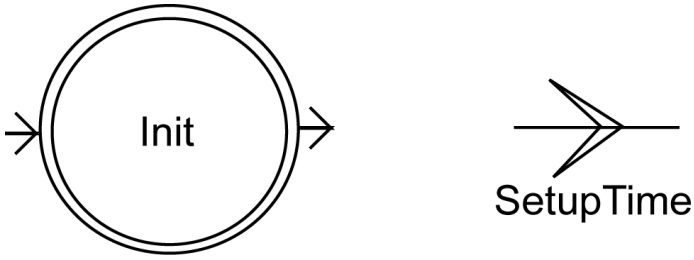
To place the Start condition:

1 **In the main library dialog box, under the category “State machine”, scroll down the list until you find “*condition*”, then double-click it.**

2 **Move the cursor to where you want to place the symbol (i.e. to the right of the stateinit symbol as shown in the diagram below), then click.**

3 **The *Edit the value* dialog box opens. Type “`SetupTime`” in the field, then click OK.**

Your scheme should now look like this:



If it doesn’t, select the incorrect symbol by clicking next to it, then delete it by pressing the *Delete* button on your keyboard, and redo steps 1 to 3 above.

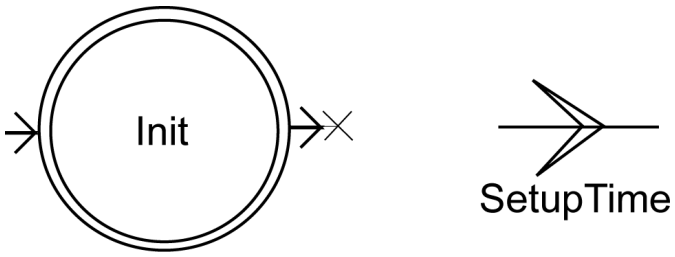
You are now going to wire the initial state and the SetupTime condition together. This forms a logical link between the two symbols. All symbols have pins to which you connect wires.

3.2.5.3 Wiring Symbols

To wire the stateinit and condition symbols together:

1 **Select wiring mode by clicking** **. The cursor changes to a crosshair, indicating that it is in wiring mode.**

2 **Place the cursor next to the right arrow on the right of the stateinit symbol. This is its output pin. The crosshair snaps onto the pin when it comes into snapping distance. An x indicates the point to which the crosshair is snapped, as shown in the following diagram:**

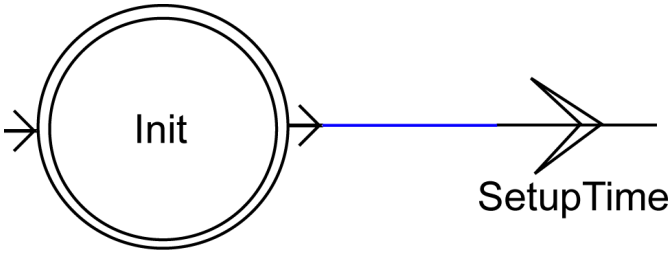


3 **Click when the crosshair is snapped to the pin. ST-Realizer will now draw a wire that follows the cursor.**

4 **Move the cursor to the line on the left of the SetupTime condition symbol. This is its input pin. Click when the crosshair is snapped onto the condition symbol's pin.**

5 **Right-click the mouse or press the ESC key or click** **to finish wiring. The two symbols are now connected by a wire.**

Your scheme should now look like this:



If it doesn't:

1 **Select the wire by clicking it.**

2 **Delete the wire by pressing the Delete button on your keyboard, and redo steps 1 to 6 above.**

3.2.5.4 Placing the Next State

To place the state, HeatingOFF:

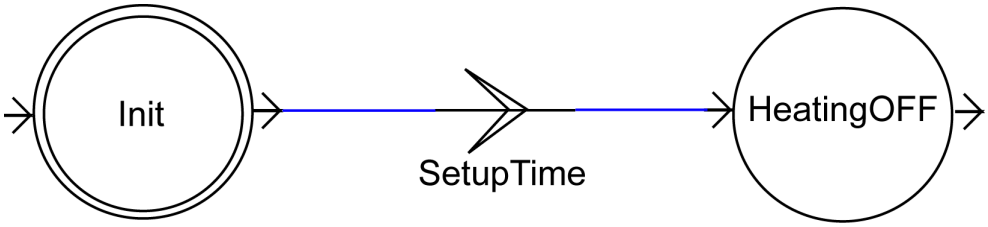
1 **In the main library dialog box, under the “State machine” category, scroll down the list until you reach *state*, then double-click it.**

2 **Move the cursor to where you want to place the symbol, then click. Place the symbol to the right of the state symbol, as shown in the diagram below.**

3 **The *Edit the value* dialog box opens. Type `HeatingOFF` in the field, then click OK.**

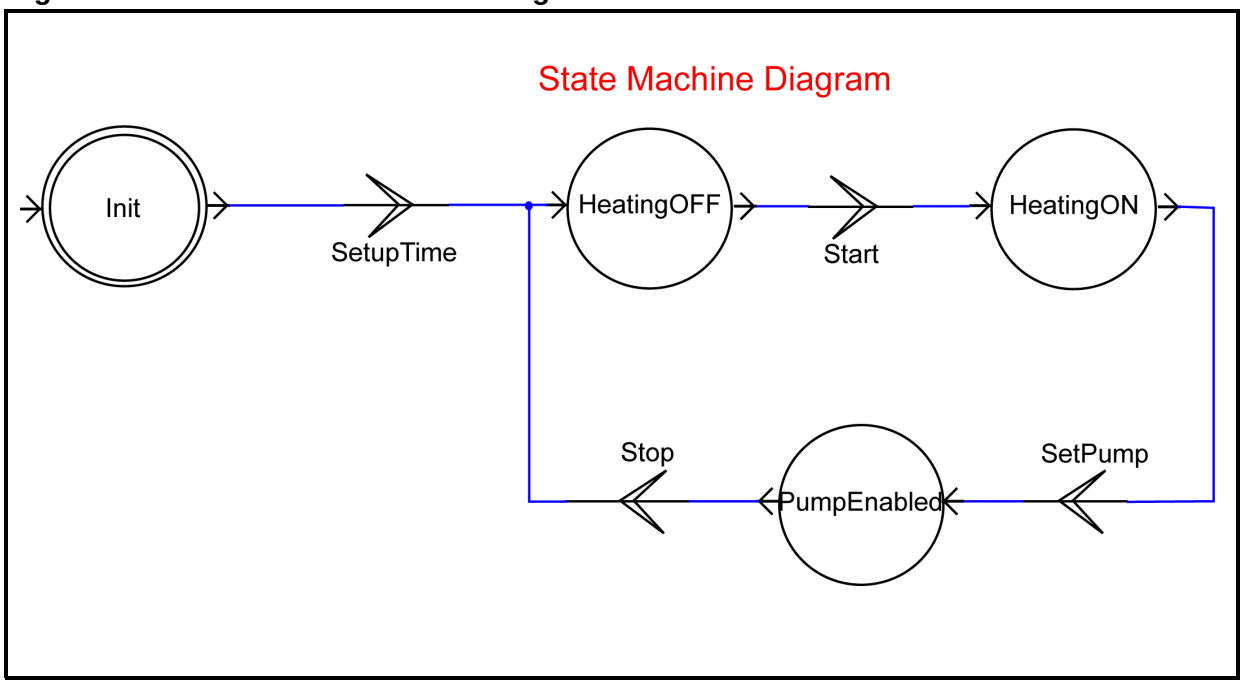
4 **Now, wire the SetupTime condition to the new state symbol as explained above.**

Your scheme should now look like this



Now design the rest of the state machine diagram, so that it appears as shown in Figure 1. The following paragraphs contain hints and explanations to help you.

Figure 1 The final state machine diagram.



3.2.5.5 Designing the Complete State Machine Diagram—Hints and Explanations

Reminder: *The application manages a **heating control system**. The ambient temperature is periodically measured, filtered and compared with a preset value. When the measured temperature is lower than the control value the heating system is (re)started while the pump speed is adjusted proportionally to the temperature differential. A pilot LED lights up to show that the heating is on. When the measured temperature exceeds the control value, the heating system is stopped.*

When the application is started, the initial state, Init, triggers a counter that allows enough time for the system to measure and filter the actual temperature value. Once this time has elapsed, the SetupTime condition is met, and triggers the HeatingOFF state, while simultaneously evaluating the temperature inputs. If the temperature differential between the actual temperature and the setpoint temperature is larger than a preset constant, a new condition called **Start** is met and a signal equal to 1 is sent. This signal triggers the transition between the HeatingOFF state and the HeatingON state, meaning that the heating system is powered and the pilot LED is switched on. To create the HeatingON state we must place a **state** symbol, to which we assign the name value **`HeatingON`**, and then wire it to the Start condition, as shown in Figure 1.

Now, the pump should operate in such a way that its speed is proportional to the differential observed between the actual temperature value and a preset temperature value. In order to do this, the pump must be enabled prior to operation. The system must change to the ***PumpEnabled*** state. A new condition symbol, ***SetPump***, is needed, that will trigger the transition between the HeatingON state and the PumpEnabled state.

Similarly, another condition, ***Stop***, will ensure the transition between the PumpEnabled state and the HeatingOFF state.

You can see that the same symbol (“condition”) is to be used four times (the “state” symbol itself is to be used three times).

When one symbol already exists in a scheme, you can copy it rather than selecting it from the library. Since you are now going to place the Start condition symbol, you can copy it from the SetupTime condition symbol:

1 **Click just next to the SetupTime condition symbol to select it.**

2 **Click** **.**

3 **Click where you want to place the copy (beside the HeatingOFF state symbol, as shown in Figure 1).**

4 **Double-click the name of the symbol. The *Edit the value* dialog box opens. Type `Start`, the name of the new condition, then click OK.**

5 **For the HeatingON state symbol, select and copy the HeatingOFF state symbol using . Place it beside the Start condition symbol. Double-click the name of the symbol and enter the new value `HeatingON`, as shown in the diagram.**

6 **Create the SetPump condition in the same manner that you created the Start condition. Note that in the State Machine diagram, the SetPump condition symbol is in the opposite direction to the Start condition symbol. You can change its direction using the mirror function, by clicking on , or by rotating it to the required position by clicking twice.**

7 **For the PumpEnabled state symbol, select and copy the HeatingON state symbol using . Place it at the bottom of the State Machine diagram. Double-click the name of the symbol and enter the new value `PumpEnabled`, as shown in the diagram.**

8 **For the Stop condition symbol, select the select the SetPump condition symbol, click to copy it, place the new symbol at the left of the PumpEnabled state symbol, then enter the value `Stop`.**

The next step is to wire the HeatingON state symbol, the three new condition symbols and the PumpEnabled state symbol. Look at the state machine diagram to see how these are wired together.

In your scheme, make sure wiring mode is selected by clicking and wire the symbols.

Note:

Auto wiring and Auto reroute are options available in the ***Option→Environment→Wiring*** menu path. These options create corners and reroute wires across the shortest path automatically. If you prefer to connect the wires another way, you can deactivate them:
1 On the **Options** menu, select **Environment.**
2 In the **Environment Options** dialog box click the **Wiring** tab.
3 Click the **Auto wiring** and **Auto reroute** check boxes. When these are empty, Auto wiring and Auto reroute are deselected.

To place a heading in the diagram:

1 **Click anywhere in the diagram area with the right mouse button.**

A popup menu displays.

2 **Select the New/Attribute options**

The ***Create an Attribute*** dialog box opens.

3 **Specify the following options:**
TAG = TXT,
Value = “State Machine Diagram” (enter text)
Visibility = Tag checkbox left unchecked

4 **Place the heading where you want, by dragging it**

5 **Click OK.**

You’ve just drawn the state machine, which is part of the main scheme of the heating control application.

The next section will explain how to draw the rest of the heating control application. Since you should now be used to placing, editing and wiring symbols, the descriptions that follow will explain what symbols you will place, with what values and why the symbols are there. They **will not** include details on how to place, edit and wire the symbols that are there. Table 2 shows a summary of the editing functions that you can use to create the remaining diagrams.

Table 2 Summary of Editing Functions

To do this:	Do this:
Select a symbol	Choose the select mode by clicking on the toolbar. Keeping the left mouse button pressed, drag a box around the symbol, or click near the symbol with the left mouse button.
Move a symbol	Select the symbol then drag-and-drop it where you want to place it.
Copy a symbol	Select the symbol. Click on the toolbar. Click where you want to place the copy.
Change the name of a symbol	Double-click the name that is currently displayed with the symbol. The *Edit the Value* dialog box opens. Type the new name. Click OK.

To do this:	Do this:
Delete a symbol	Select the object. Press the Delete key on your keyboard.
Wire two symbols together	Select the wiring mode by clicking . Place the cursor next to the output pin on the first symbol. A crosshair snaps onto the pin when it comes into snapping distance. Move the cursor to the appropriate pin of the other symbol. This is an input pin. Click when the crosshair is snapped onto the symbol's pin. The two places where you clicked are now connected by a wire. Click the right mouse button or press the ESC key to finish wiring.

3.3 Completing the Heating Control Application

The State Machine Diagram explained the relationship between the state of the heating control system and certain conditions. The next step is to link each state with an action, and to define the rules by which each condition is, or is not, met.

To do this, you need to draw the other parts of the main scheme and subschemes that, together, make up the heating control application—specifically:

- ***External Input conditions***, that are transmitted to the application from external sources, such as the measured (actual) temperature. This function will be part of the **Main Scheme**.
- ***Internal input conditions*** that control the Setup Timer and enable the pump. These functions will be part of the **Main Scheme**.
- ***External actions***, that are output from the application, such as putting on the pilot LED, or controlling the speed of the pump. This function will be part of the **Main Scheme**.
- An ***internal event-driven action*** such as the periodic activation of the temperature acquisition and filtering process. This function will be part of a **subscheme** called **`filter.sch`**, which we will learn how to create in Section 3.3.10 on page 35.

3.3.6 What differentiates a main scheme from a subscheme?

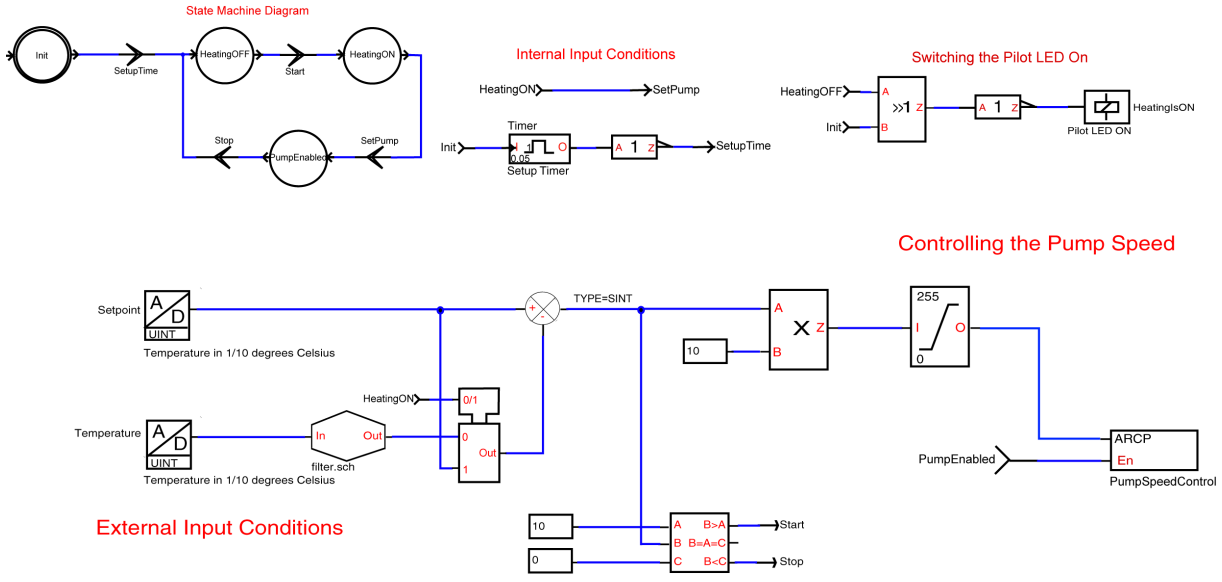
Each diagram (meaning a collection of symbols wired together) in a scheme is analogous to a sub-routine in a program. The main scheme is, if you will, the conceptual core of your application. It pays to keep things simple in the main scheme, and to only include diagrams that represent the large-scale, sequential running of the application.

All of the highly detailed acquisition, filtering and comparison programs are better put into subschemes so that the application will be easier to de-bug or update later.

There are two good guidelines to keep in mind when deciding when to put something into a subscheme, rather than in the main scheme:

- Size and complexity of the diagram - the larger and more complex the diagram, the more reason to put it into a subscheme by itself,
- If the function that the diagram performs runs in parallel to other functions (for example, a sampling program that periodically reads external inputs continuously), **it must be put in a subscheme**.

The entire main scheme of the heating control application is shown below. Only one subscheme—for the internal event-driven action described above—will be created after we finish the main scheme.



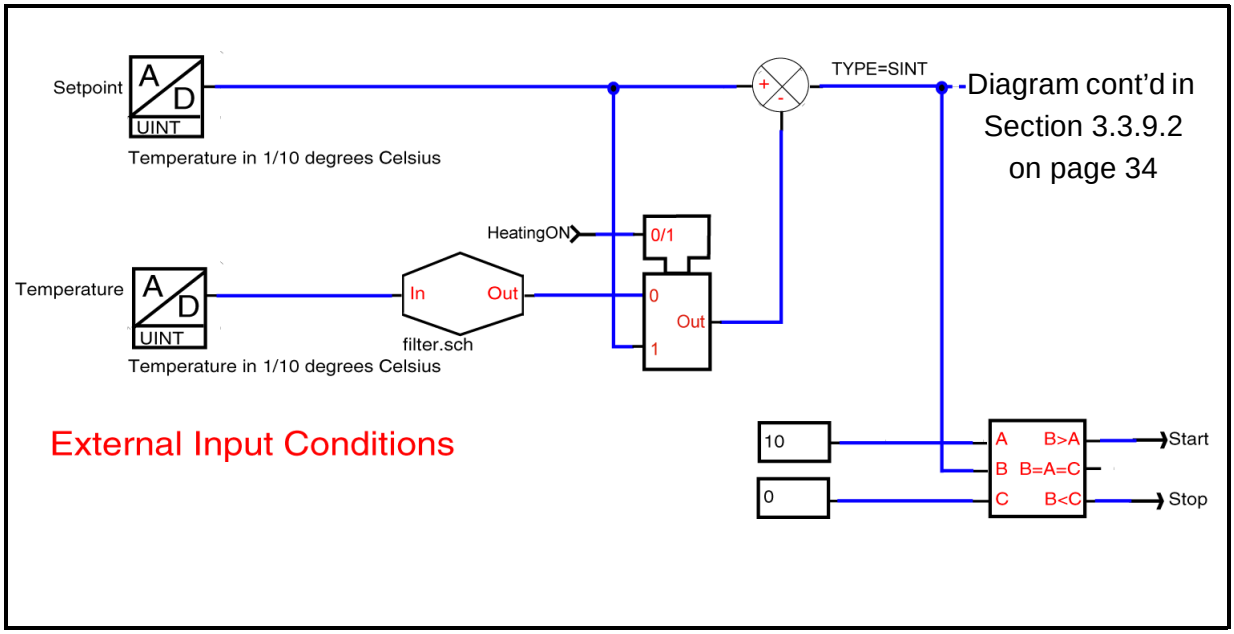
We have already completed the State Machine Diagram (top left). The remainder of the main scheme diagrams are described in the following paragraphs.

3.3.7 External Input Conditions

The external input conditions proceed from two inputs:

- The control temperature (“Setpoint” in the diagram shown below).
- The measured temperature (“Temperature” in the diagram shown below).

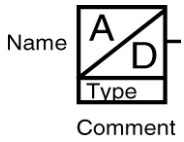
The above external input conditions are indicated on the input pins on the microcontroller. Each time an input signal changes, it is processed, and the appropriate condition (Start or Stop) resulting from the comparison is signalled to the state machine by changing the condition value to 1. The processing is designed as follows:

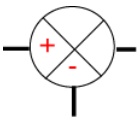


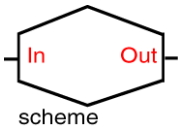
Draw the above diagram on your heater main scheme, with the help of the following tables that describe the symbols you’ll place, what they do and the values entered with them.

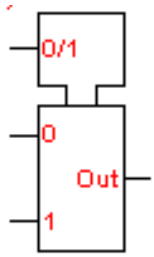
Don’t forget to wire your diagram appropriately.

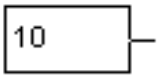
3.3.7.1 Symbols

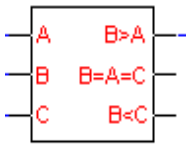
Symbol	Functional Category	Name	Values
	Input and output	Symbol: **adc**	There are two occurrences of this symbol in the diagram: Comment = As shown in diagram Name = SetPoint and Temperature. Type = UINT
Description This is an analog to digital converter input symbol. The NAME value connects it to a hardware port (see “Connecting Hardware Ports and Peripherals” on page 38). The TYPE value is used to define the variable type (UBYTE, SBYTE, UINT, SINT or LONG). The two instances have the Type UINT, which is unsigned integer. The comment value is used in the report file (see “Analysing and Generating Program Code” on page 44).			

Symbol	Functional Category	Name	Values
	Mathematical	Symbol: **sub2**	None
Description This is a two input subtractor, with type inheritance. OUT = IN1 - IN2. It subtracts the actual temperature from the Control Temperature. For details on type inheritance, see “Type Inheritance” on page 204.			

Symbol	Functional Category	Name	Values
	Hierarchical Sheet	Symbol: **sss1_1**	Scheme = Name of the related subscheme file (*filter.sch*, in the example).
Description This is a subscheme connection symbol. It represents a process that is contained in a sub-scheme. In this case, the subscheme has one input (“In”) and one output (“Out”). By using Portin and Portout symbols within the subscheme the connection is made between the pins of this symbol and the subscheme. This subscheme is described in Section 3.3.10 on page 35.			

Symbol	Functional Category	Name	Values
	Logic	Symbol: **mux1**	None.
Description This is a single-output multiplexer, where an input condition determines whether the output value should be equal to input 0 or input 1. For example, when the input condition equals 1, the output value will be the input value from line 1.			

Symbol	Functional Category	Name	Values
	Constant	Symbol: **constw**	Value = 10 There are two instances in this diagram, one in the ARTimer subscheme diagram, and another one in the Filter subscheme diagram.
Description This is a word constant value symbol that inputs the value 10.			

Symbol	Functional Category	Name	Values
	Conversion	Symbol: **comp**	None
Description This is a multi purpose comparator. It returns three bit values that depend on the three inputs. B>A = 1 when B is greater than A. B = A = C = 1 when B is equal to A and C. B < C = 1 when B is smaller than C. There is one occurrence of this symbol. It indicates the Start condition, when this condition is met, by outputting 1. This condition is met when the differential between the control temperature and the actual temperature is greater than 1 degree C. It indicates the Stop condition (output=1 on this pin) when the differential between the control temperature and the actual temperature is negative.			

Symbol	Functional Category	Name	Values
	State machine	Symbol: **stateout**	Name = HeatingON Several additional occurrences of this symbol can be found in other main scheme diagrams.
Description This is an example of a state output symbol. State output symbols are connected to state symbols by their names. When the system switches to the specified state, the processing that follow the state output symbol is performed. In the example, the processing simply consists of enabling the pump.			

Symbol	Functional Category	Name	Values
	State machine	Symbol: **statein**	There are two occurrences of this symbol in this diagram: Name = Start Name = Stop Another instance of this symbol can be found in the Internal Input Condition diagram.
Description These are state input symbols, that connect to condition symbols in the state machine that have the same name. For example, when the bit value 1 is received from the B>A output of the comparator, it signals the Start condition by outputting the value 1 to the state machine.			

3.3.7.2 Wires

Occasionally, you must attach data type attributes to wires to control the outputs. For example, you must attach attribute TYPE = SINT to the wire connected to the output pin of the **sub2** symbol in order to be able to evaluate both positive and negative temperature differentials.

To attach a type to a wire, proceed as follows:

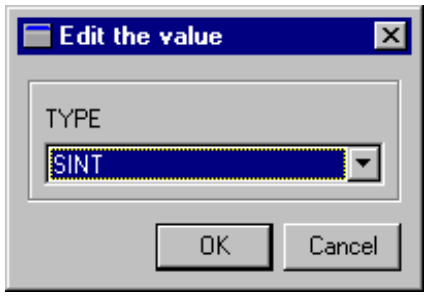
1 **Click the wire with the left mouse button.**

2 **Click the right mouse button.**

A list box displays, with 3 attributes: ***Type, Init, Label***.

3 **Click *Type*.**

The ***Edit the value*** dialog box opens, with a scrolling list of data types to choose from.



4 Select the type, `SINT`, and click OK.

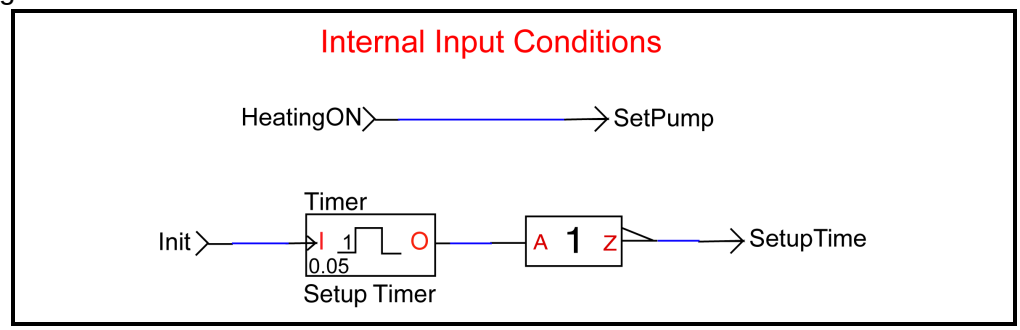
`SINT`, stands for *Signed Integer*.

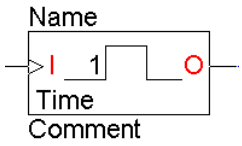
3.3.8 Internal Input Conditions

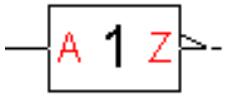
There are two internal input conditions that must be met:

- Enough time must pass to allow temperature values to be acquired and filtered before the temperature differential is measured. When this has happened, the SetupTime condition is met.
- Once the HeatingOn state is active, the pump must be enabled prior to being put into operation by activating the SetPump condition.

The diagram is shown below:



Symbol	Functional Category	Name	Values
	Time related	Symbol: **timf**	Name = Timer Time = 0.05 Comment = Setup Timer
Description This is a simple timer. The output signal is 1 until the time specified has elapsed, and then the output becomes 0. This symbol allows enough time for the application to acquire and filter temperature values.			

Symbol	Functional Category	Name	Values
	Logic	Symbol: **inv**	None. This symbol also occurs later in the application.
Description This is a multi-type bit-wise inverter. The 0 output from the timer is reverted to 1, triggering the SetupTime condition			

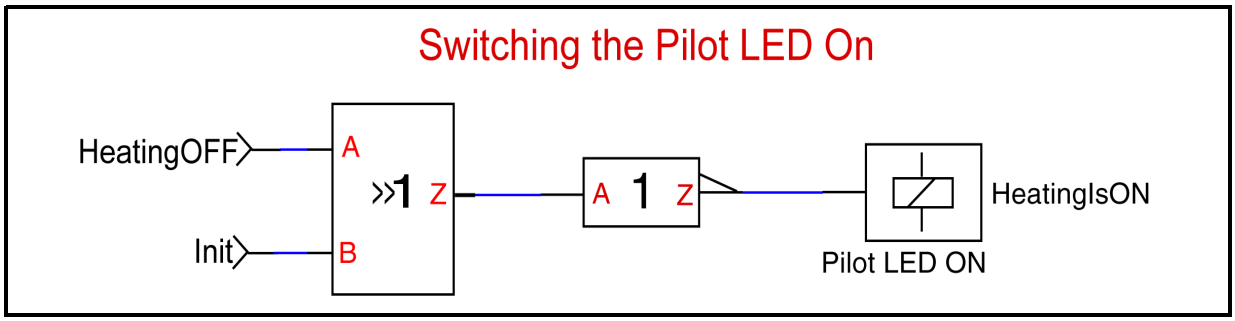
3.3.9 External Actions

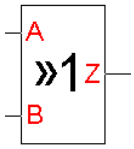
External actions generate output signals relating to the current state in the state machine. The output signals are:

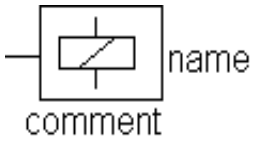
- The signal to switch on the Pilot LED.
- A PWM signal generated by the Auto-Reload Timer to control the speed of the pump.

3.3.9.1 Pilot LED Monitoring

As long as the state of the system is HeatingOFF, the Pilot LED should be OFF. The corresponding diagram is:



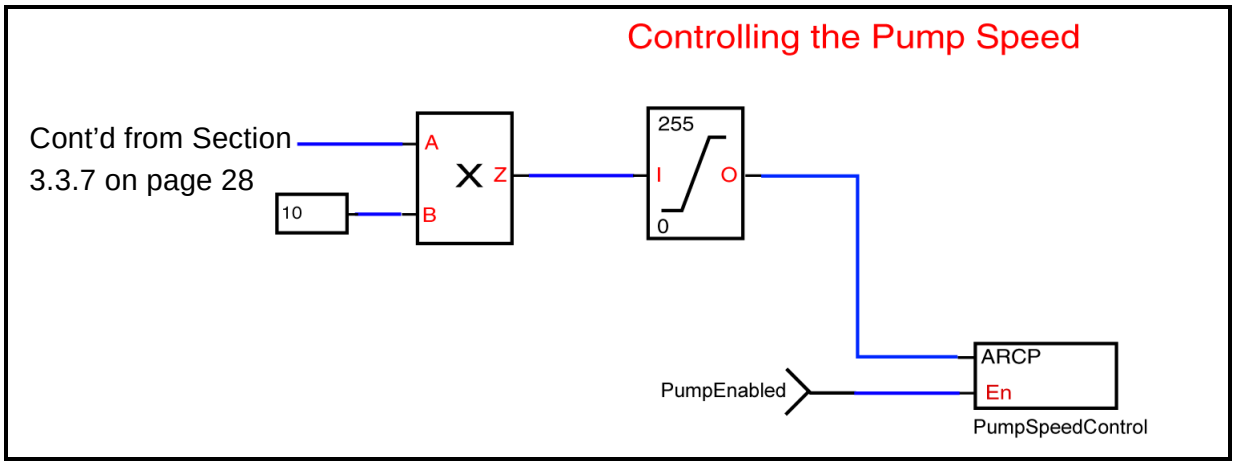
Symbol	Functional Category	Name	Values
	Logic	Symbol: **or2**	None.
Description This is a two-value binary OR function symbol with type inheritance. If either the HeatingOFF or Init states are active, the Pilot LED will be remain off. However, if neither of these states is active, the LED will be ON.			

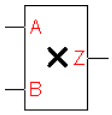
Symbol	Functional Category	Name	Values
	Input and output	Symbol: **digout**	Only one instance in the example: Name = HeatingIsON Comment = Pilot LED ON
Description This is a digital output symbol. The name value is used for connecting this symbol to the microcontroller output ports (see Section 3.3.11 on page 38). The Comment value is used in the report file.			

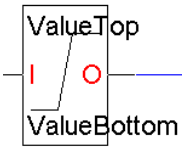
3.3.9.2 Pump Speed Control

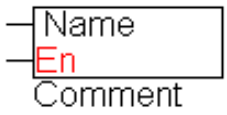
A Pulse Width Modulation (PWM) signal generated on the OCMP1_B output pin (TBOC1LR Timer B port, for the ST72212G2) ensures pump speed control. For this, the ARCP compare register is loaded with a value between 0 and 255 that reflects the temperature differential calculated by comparing actual and control temperature inputs (see the “external input conditions” diagram above).

The pump speed control diagram in the main scheme is as follows:

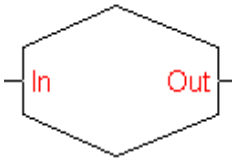


Symbol	Functional Category	Name	Values
	Mathematical	Symbol: **mul**	None.
Description This is a 2-input multiplier symbol with type inheritance. Z=A*B. Type for A and B can be any type from UBYTE through LONG. Type for Z is the largest of A or B types. For details on type inheritance, see “Type Inheritance” on page 204.			

Symbol	Functional Category	Name	Values
	Mathematical	Symbol: **limf**	ValueTop = 255 ValueBottom = 0
Description This is a fixed-value limiter symbol. The output will not be larger than the top value and not smaller than the bottom value. Type of O = type of I = any type from UBYTE through LONG.			

Symbol	Functional Category	Name	Values
	Input and output	Symbol: outputlatch	Name = ARCP Comment = PumpSpeedControl
Description This is an output symbol. The NAME value connects it to a hardware port (see Section 3.3.11 on page 38). Any type from BIT through LONG can be used.			

3.3.10 The Creation of a Subscheme for an Internal Event-Driven Action

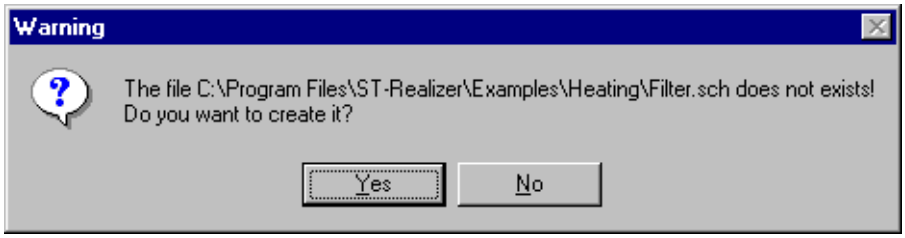
In Section 3.3.7, we made reference in the main scheme to a subscheme for an internal event-driven action. This particular action controls the periodic activation of the temperature acquisition and filtering process and reflects a ***timed interrupt execution condition***—that is, an event periodically triggered by a timer interrupt. Recall from Section 3.3.6 that all events that need to be executed in parallel, or independently from, the main scheme, must be placed in subschemes. In our tutorial application, the subscheme attached to this action (*filter.sch*) groups code that executes as an interrupt service routine. This single-input, single-output subscheme is represented by the **sss1_1** symbol 

, embedded in the main scheme diagram where the input pin of the symbol corresponds to the input in the subscheme, and the symbol's output pin corresponds to the output of the subscheme. Of course, subschemes can have various numbers of inputs and outputs, and can be represented by symbols of type "**sssx_y**", where x = the number of inputs and y = the number of outputs.

To create a subscheme:

1 **Double-click on the above symbol in the main scheme diagram.**

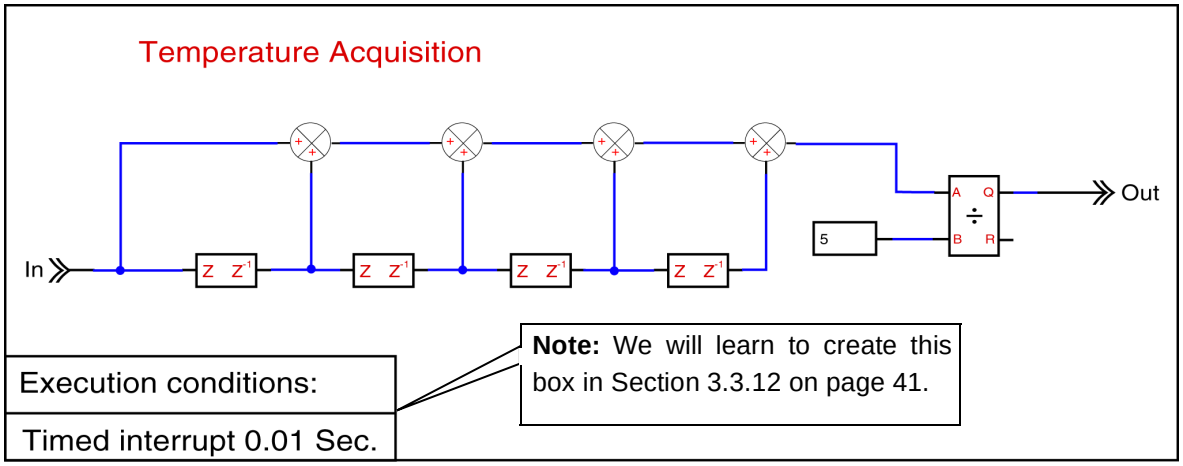
2 **A window (shown below) will appear asking if you wish to create the new subscheme. Click OK.**



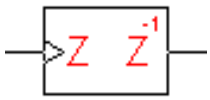
A new blank scheme will appear.

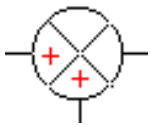
3.3.10.1 Temperature Acquisition and Filtering

The subscheme diagram is as follows:



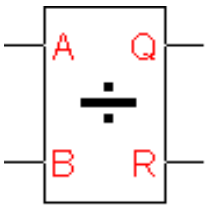
Symbol	Functional Category	Name	Values
	Hierarchical sheet	Symbol: **portin**	Label = identifier used to make the connection between this subscheme and the parent scheme. Must be the name of the input pin in the **sss*1_1*** symbol that can be found in the main scheme diagram. In the example, LABEL = “In”.
Description This symbol is used to connect an input pin of a subscheme symbol (“sssx_y”) from the parent scheme with the corresponding subscheme. This symbol is homologous to the portout symbol which describes an output pin in a subscheme symbol.			

Symbol	Functional Category	Name	Values
	Logic	Symbol: **loopdel**	None.
Description In other words the output value of this symbol is the value obtained during the previous polling loop. This symbol acts as a one-loop delay symbol. Type of Z^{-1} = type of Z = any type from BIT through LONG. In the example, this symbol is used 4 times: this means that temperature measurements are performed 5 times, one polling loop after the other. The five temperature values are then averaged to smooth the effect of temporary variations.			

Symbol	Functional Category	Name	Values
	Mathematical	Symbol: **add2**	None

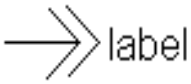
Description

This is a two input adder, with type inheritance. OUT = IN1 + IN2. For details on type inheritance, see “Type Inheritance” on page 204.

Symbol	Functional Category	Name	Values
	Mathematical	Symbol: **div**	None.

Description

This is a 2-input divider symbol with type inheritance. Q = A/B, R = A modulo B. Type for A and B can be any type from UBYTE through LONG. Type for Q and R is the largest of A or B types. For details on type inheritance, see “Type Inheritance” on page 204.

Symbol	Functional Category	Name	Values
	Hierarchical sheet	Symbol: **portout**	Label = identifier used to make the connection between this subscheme and the parent scheme. Must be the name of the output pin in the **sss*1_1*** symbol that can be found in the main scheme diagram. In the example, LABEL= “Out”.

Description

This symbol is used to connect an output pin of a subscheme symbol (“**sss*x_y***”) from the parent scheme with the corresponding subscheme. This symbol is homologous to the **portin** symbol which describes an input pin in a subscheme symbol.

3.3.10.2 Leaving a Subscheme

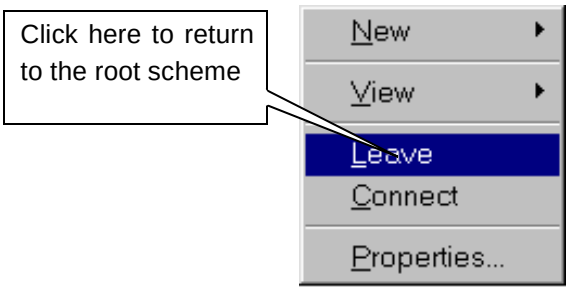
If you are in a subscheme, you can return to the root scheme by either of these actions:

- **Click Close in the File option of the main menu, or,**
- **Click with the right mouse button any blank position in the subscheme.**

A popup menu opens with five options.

Select *Leave.*

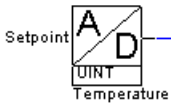
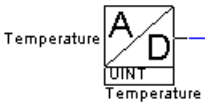
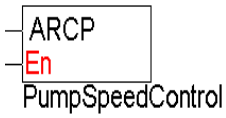
You will be returned to the root scheme.



3.3.11 Connecting Hardware Ports and Peripherals

Once you have finished drawing your application schemes, the next step is to connect the application inputs and outputs to the ST6 or ST7 microcontroller input and output hardware ports, or peripherals.

In our heating control application, the input/output symbols are:

Symbol	Symbol Name	Type	Function in Diagram
	**adc**	**Analog Input**	Control temperature acquisition, in External Input Conditions diagram.
	**adc**	**Analog Input**	Actual temperature acquisition, in External Input Conditions diagram.
	digout	**Output**	HeatingIsON, in External Actions diagram.
	**outputlatch**	**Output**	Loading ARCP Compare Register, in External Actions diagram.

Therefore, there are four input/output symbols, and they need to be connected as shown in Table 3[(1)]. The four input/output symbols represent two types of devices:

- Three **ports**, namely Setpoint (PC.4), Temperature (PC.5) and HeatingIsON (PB.0).
- One **on-chip peripheral**, ARCP (Timer B), to be activated in Auto Reload mode with PWM generation via the TBOC1LR Timer B (Output Compare 1) Register.

Table 3 Application—Microcontroller Connections.

Application Symbol:	Microcontroller Pin:
SetPoint (adc)	PC.4, 8-bit analog input
Temperature (adc)	PC.5, 8-bit analog input
HeatingIsON (digout)	PB.0, Push-pull output
ARCP (outputlatch)	TBOC1LR Timer B Register

To make these connections, follow the instructions for connecting a port device for the Setpoint, Temperature and HeatingIsON symbols and the instructions for connecting an on-chip peripheral for the ARCP symbol.

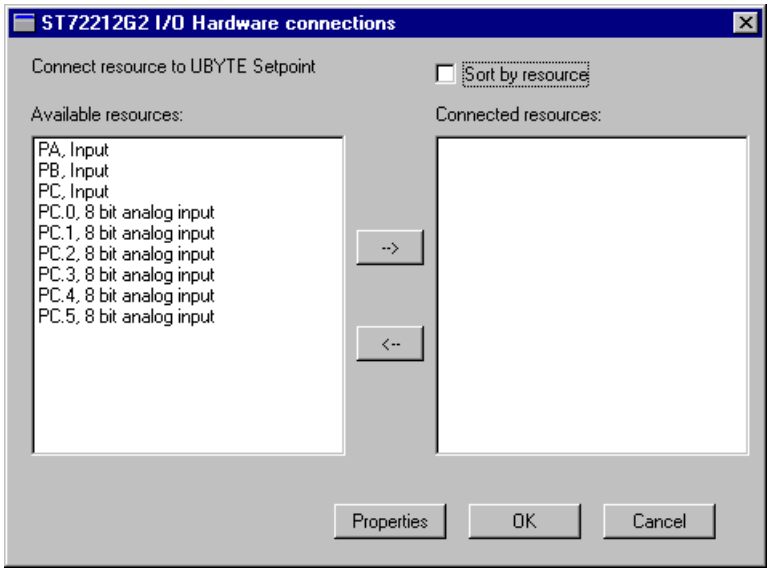
3.3.11.1 Connecting a port device

You're now going to connect the SetPoint input:

1 **Double-click the SetPoint adc symbol.**

The I/O Hardware connections dialog box opens listing the available pins for the type of connection you want to make:

You are going to connect the first adc to PC.4, 8-bit analog input.



1 Pin connections depend upon the application and target microcontroller. For this tutorial application, and our target microcontroller, ST72212G2, we have specified the necessary pin connections. In order to correctly create your own application, you will need to be familiar with the target microcontroller for which it is intended.

2 In the *Available resources* list, double-click the entry `PC.4,8-bit analog input` and then click OK.

Now connect the **Temperature** and **HeatingIsON** inputs in the same way.

When you connect port devices, the dialog boxes automatically display input or output pins depending on the type of symbol. There is no need for you to configure the ports as input or output, prior to connecting them.

However, for on-chip peripherals, such as the **Timer B** on the ST72212G2, you need to specify some hardware settings before the appropriate resource appears in the dialog box. For instructions on how to specify these settings, see below.

3.3.11.2 Connecting and Specifying Hardware Settings for an On-Chip Peripheral

To connect the on-chip peripheral, Timer B, proceed as follows:

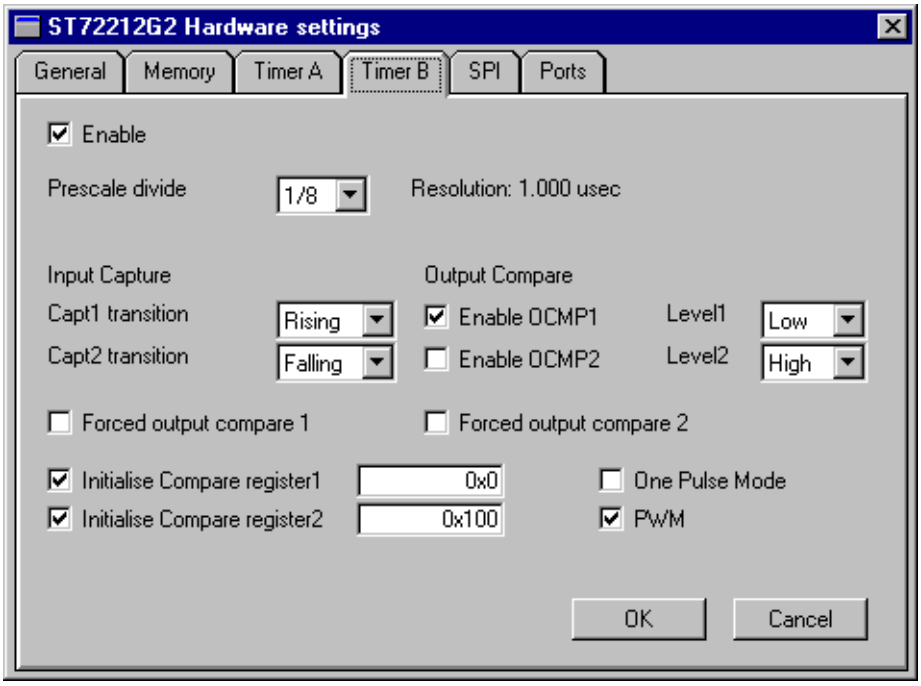
1 Click the Hardware Settings entry in the Project menu.

or:

Click the Properties button in the I/O Hardware connections dialog box.

2 In the Hardware settings dialog box, click the Timer B tab.

The Timer B dialog box opens.



3 At the top of the Timer B Dialog box, click the Enable box.

This action permits the application use of the Timer B peripheral and allows access to the interrupts and control registers of the peripheral.

4 Set the following parameters:

- Prescaler division = 1/8.
- Input Capture: Capt1 transition = Rising, Capt2 transition = Falling.
- Output Compare: Click the Enable OCMP1 box and set Level 1 = Low and Level 2 = High.
- Click the Initialise Compare register1 box and set the value to 0x0.
- Click the Initialise Compare register2 box and set the value to 0x100.
- Click the PWM box.

5 Click OK to confirm.

The dialog box closes.

The Timer B on-chip peripheral is now enabled as a PWM generating device. Consequently, a Pulse-Width Modulated signal can be generated on the OCMP1 B output pin.

6 Now that the device is correctly configured, follow the procedure in Section 3.3.11.1 on page 39 and connect the ARCP symbol (outputlatch) to the TBOC1LR Timer B register.

3.3.12 Event Control

The last thing you must do when you create an application is to assign execution conditions to those events that are represented in subschemes.

Note: Execution conditions cannot be assigned to main (root) schemes—it simply doesn't make sense to use the main scheme as an interrupt routine, for example. In addition, it is best to define execution conditions after having enabled (see page 40) any necessary microcontroller peripherals, so that these peripheral functions are available for use.

Execution conditions:
Timed interrupt 0.01 Sec.

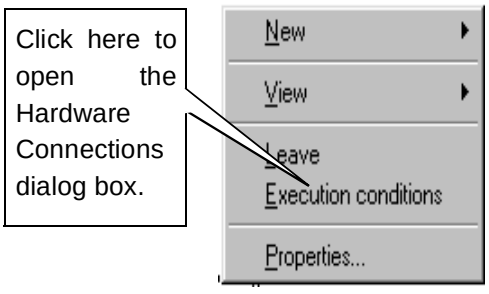
The execution conditions box, shown at the bottom left corner of the subscheme, (as seen in the Temperature Acquisition subscheme diagram on page 36) is created as follows:

1 Right-click the mouse in any blank position in the diagram.

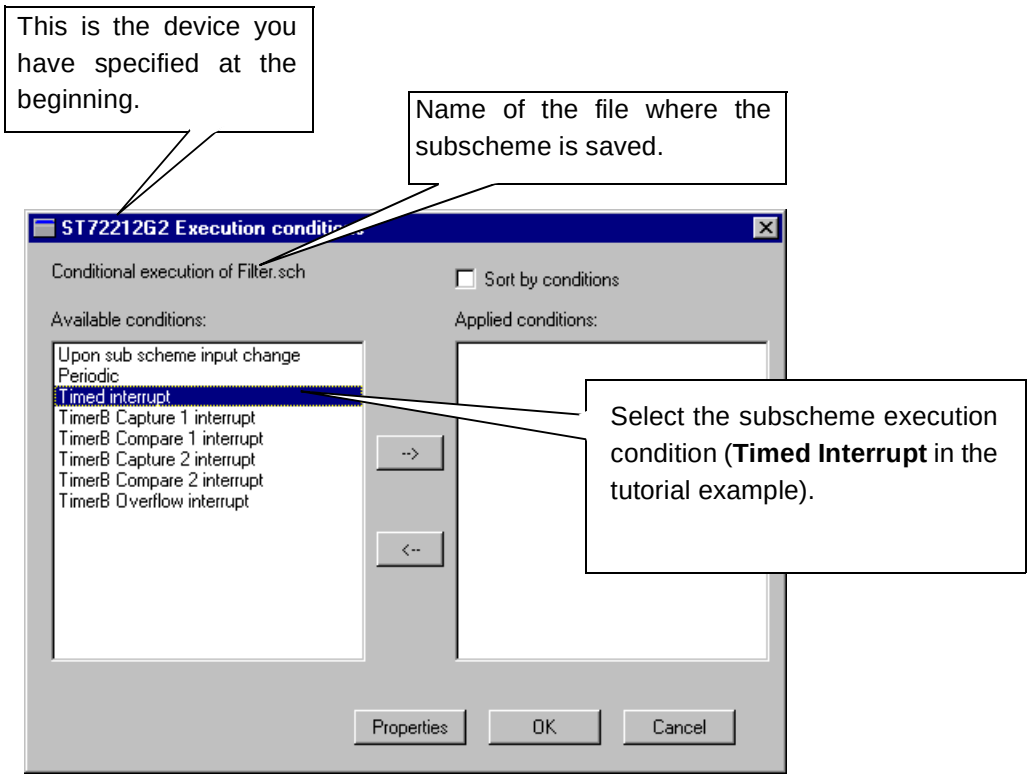
A popup menu opens with five options.

2 Select *Execution Conditions.*

The Execution Conditions dialog box for the specified target microcontroller (ST72212G2, in the example) opens. Specifying this target microcontroller was one of the first tasks you had

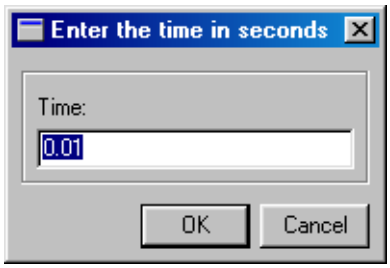


to perform (see Section 3.1.2 on page 14).



3 **In the *Execution Conditions* dialog box (see above), double-click the *Timed Interrupt* line.**

A new dialog box opens for you to specify the time period between input value acquisitions.



4 **Enter 0.01 (1/100 second) and click OK.**

5 **Click the OK button in the *Execution Conditions* dialog box.**

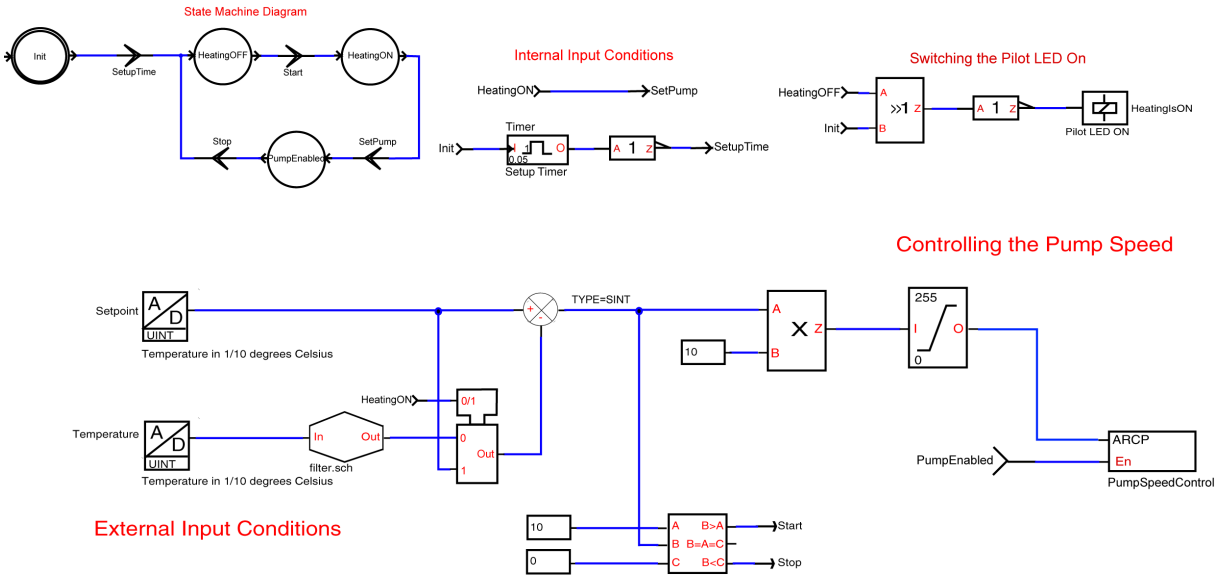
6 The ***Timed Interrupt*** event is now connected to the subscheme, with a time value of 0.01 seconds. What does this mean? It means that the code described by the Temperature Acquisition and Filtering subscheme will be executed every 1/100 of a second, after the main processing loop has been interrupted on a priority basis. The polling of the temperature capture points and the consolidation of the measured temperatures will thus be performed every 0.01 seconds.

3.3.13 Summary

You have now completed the heating control application schemes. You have learned through doing this:

- How to place and edit symbols.
- What the principal symbols do, and when to use them.
- How to wire symbols together to create a main scheme and a subscheme.
- How to connect application inputs and outputs to target microcontroller pins.
- How to define the execution conditions of a subscheme.

Recall that the main, or root, scheme for the heating control example will look like the following:



The next part of this tutorial will show you how to analyse and generate the code for the heating control application.

3.4 Analysing and Generating Program Code

In this part of the tutorial you are going to learn how to analyse the schemes you draw and generate the program code for them.

When you execute the analysis and compile, ST-Realizer analyses your scheme by creating the netlist, creating cross references, analysing and generating final code. During these phases the connectivity between symbols, the I/O assignment and variable types are checked before generating the source code. Providing no fatal errors are encountered, ST-Realizer generates a non-compiled ST6 or ST7 macro-assembler language (**.asm**) file from the scheme, which it then compiles.

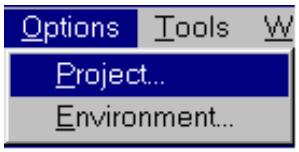
When the analysing process has been successfully completed, a report file is generated. This report file gives information about the designation of I/O pins, a list of the variables used by type and the memory space required by the application.

3.4.14 Setting the Compile Options

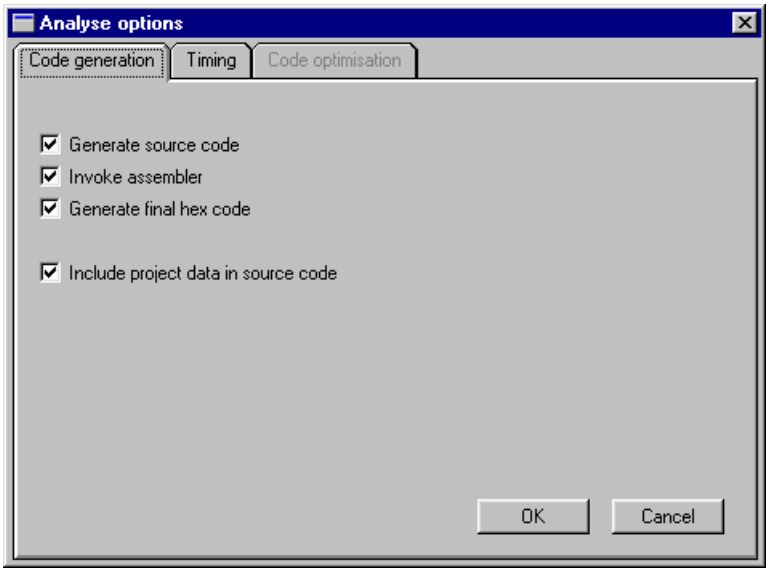
There are a number of compile options that you may set before performing an Analysis of your application. These options are discussed in detail in Section 9.2 on page 129. However, most of the default settings are suitable for our tutorial application.

To view or set the compile options:

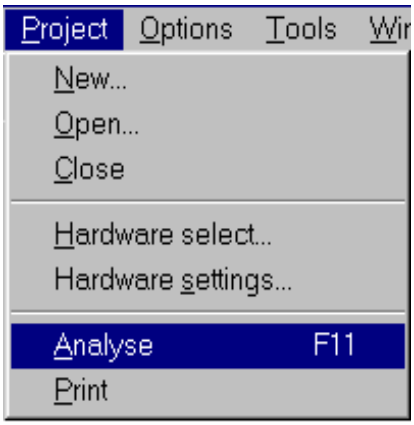
1 On the main menu, click *Options*, then *Project*...



The Code Generation dialog box in the Analyse Options window opens. The default settings are suitable for our tutorial application. **Click OK**.

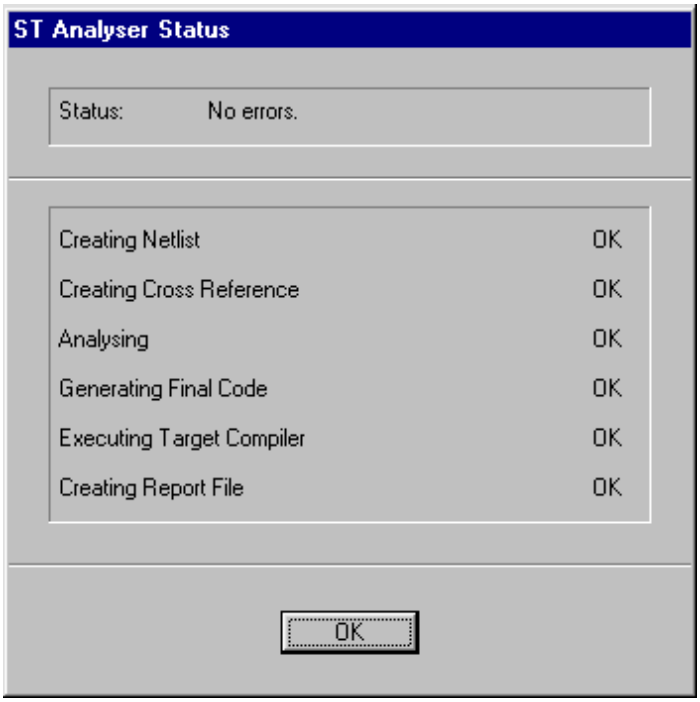


3.4.15 Executing the Analysis and Compile



You may execute the ST-Analyser from any of the application windows.

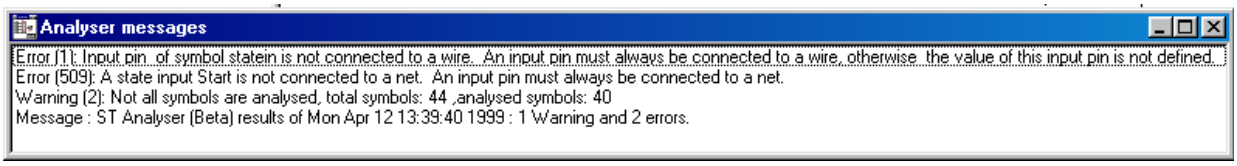
On the **Project** menu, simply click **Analyse**. The ST Analyser Status window will open and you will see the Analyse function checking your application step-by-step.



If there **are** errors:

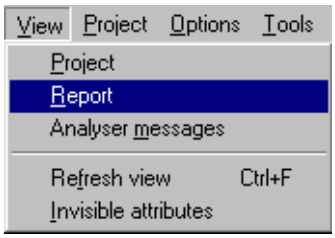
1 **A dialog box will open, showing the status of the compilation, with the number of errors and warnings. Click OK.**

2 **A descriptive list of the errors will be shown in the Analyzer messages window at the bottom of the scheme window. You can view each error by either double-clicking on its message in the window, or by clicking the previous message** **or next message** **icons on the toolbar. The area of the scheme where the error occurs will appear in the scheme window.**


Analyser messages
Error (1): Input pin of symbol statein is not connected to a wire. An input pin must always be connected to a wire, otherwise the value of this input pin is not defined.
Error (509): A state input Start is not connected to a net. An input pin must always be connected to a net.
Warning (2): Not all symbols are analysed, total symbols: 44 ,analysed symbols: 40
Message : ST Analyser (Beta) results of Mon Apr 12 13:39:40 1999 : 1 Warning and 2 errors.

3 **Correct the errors and re-compile the application by clicking on Analyse under the Project menu.**

3.4.16 Viewing the Analyse and Compile Report



Once you have analysed your scheme and compiled your program code, you can view the report generated by ST-Realizer during the analysis and compilation process. This report provides you with useful information such as the input and output connections you made, and gives an overview of how much memory is used by the application.

To see the report, on the **View** menu, click **Report**. A portion of our application report is shown below:

```
ST72212G2 Realizing Unit (V4.00) (c) 1990-98 Actum Solutions
Report file of project C:\Program Files\ST-Realizer\Examples\Heating\heating.rpf
Scheme Version   : 1.00
Report timestamp: Wed Apr 14 13:14:40 1999
Analyze results  : No errors
-----------------------------------------------------------------

Schematic dependencies and events:
-----------------------------------------------------------------
C:\Program Files\ST-Realizer\Examples\Heating\heating.sch
Scheme: C:\Program Files\ST-Realizer\Examples\Heating\Filter.sch
Event:  Timed interrupt 0.01 Sec.


ST72212G2 (DIL28) connection overview:
-----------------------------------------------------------------
Pin    Name      Alternative name   Type   I/O          Description
 1:    RESET                        (BIT   Input ),     Active low
 2:    OSCin                                ),     Oscillator In
 3:    OSCout                       (             ),    Oscillator Out
 4:    PB.7      SS                 (BIT   Input ),     Not connected
 5:    PB.6      SCK                (BIT   Input ),     Not connected
 6:    PB.5      MISO               (BIT   Input ),     Not connected
 7:    PB.4      MOSI               (BIT   Input ),     Not connected
 8:    PB.3      OCMP2_A            (BIT   Input ),     Not connected
 9:    PB.2      ICAP2_A            (BIT   Input ),     Not connected
10:    PB.1      OCMP1_A            (BIT   Input ),     Not connected
11:    PB.0      HeatingIsON        (BIT   Output),     Push-pull output
12:    PC.5      Temperature        (UBYTE Input ),     8 bit analog input
13:    PC.4      Setpoint           (UBYTE Input ),     8 bit analog input
14:    PC.3      ICAP2_B            (BIT   Input ),     Used by the application
15:    PC.2      CLKOUT             (BIT   Input ),     Not connected
16:    PC.1      OCMP1_B            (BIT   Input ),     Used by the application
17:    PC.0      ICAP1_B            (BIT   Input ),     Used by the application
18:    PA.7                         (BIT   Input ),     Not connected
19:    PA.6                         (BIT   Input ),     Not connected
20:    PA.5                         (BIT   Input ),     Not connected
21:    PA.4                         (BIT   Input ),     Not connected
22:    PA.3                         (BIT   Input ),     Not connected
23:    PA.2                         (BIT   Input ),     Not connected
24:    PA.1                         (BIT   Input ),     Not connected
25:    PA.0                         (BIT   Input ),     Not connected
26:    TEST                         (              ),   Test mode pin
27:    Vss                          (              ),   Ground
28:    Vdd                          (             ),    Power Supply
```

The next, and final part of this tutorial will to describe how to simulate and fine-tune your application using ST-Simulator.

3.5 Simulating and Fine-Tuning Your Application

Once you have designed and analysed your application, you can use ST-Simulator to simulate its behavior, generate and view input signals, monitor signals that are generated by your application, and fine-tune it if necessary. In this part of the tutorial, you're going to:

- Create a simulation environment file, which defines the environment in which you'll simulate your application.
- Add adjusters to your simulation environment file, which enable you to generate and view the signals that are input to your application.
- Add probes to your simulation environment file, which enable you to view the signals that are generated by your application.
- Run the simulation.

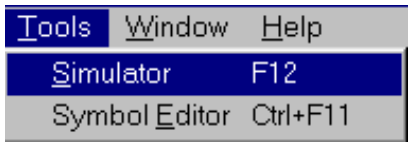
3.5.17 Creating the Simulation Environment File (.sef)

The first step in simulating your application is to create its simulation file. Simulation files are based on the application schemes and the target device, and are created in ST-Simulator.

To create the heating control simulation environment file:

1 Make sure that the heating control application is open.

2 On the *Tools* menu, click *Simulator*.



The ST-Simulator window opens.

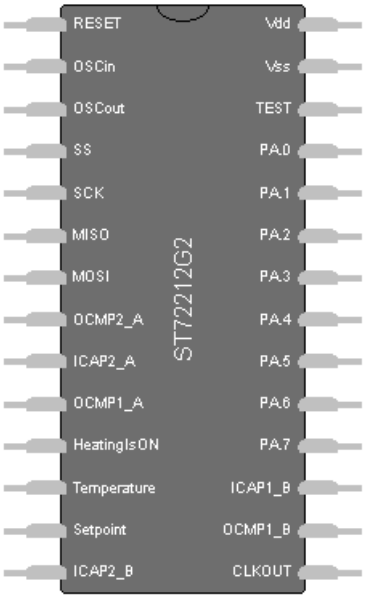
3 On the *File* menu, click *New→Simulation environment.*

The Create a New File dialog box opens, letting you assign a name for the new simulation environment.

Type the name of the .sef file: `heating.sef`. Click *Save*.

You've now created the simulation environment file for the heating control application.

A pin level drawing of the ST72212G2 chip appears as shown at right. Note that where the pins have been connected to application input/output functions, the pin names have been replaced by the function names.



If you double-click on the pin level drawing, you will open a copy of your root scheme diagram as another window. You can also open subschemes by double-clicking on the subscheme symbols.

You can use both the pin level drawing and the scheme diagrams to help you run the simulation—both views are useful in evaluating whether your application is running as you wish.

To see all views at once, under the *Window* menu, select *Tile*.

3.5.18 Connecting Probes and Adjusters

You are now going to connect probes and adjusters to your scheme, so that you can adjust input values to the application and view output values. This enables you to experiment with and fine-tune your application. There are four types of adjusters and three types of probe available, and these are described in detail in Sections 10.2 and 10.3, respectively.

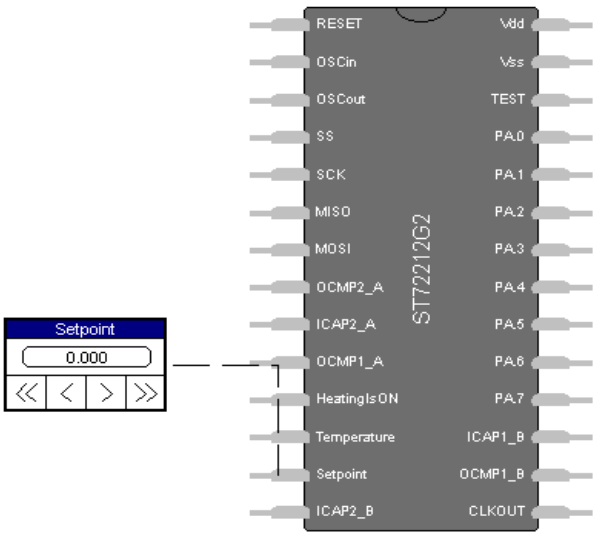
Connecting an Adjuster

In our application, recall that there are two temperature inputs: a Setpoint temperature, which is the desired temperature for the system, and the Actual temperature, which is read and

filtered by a temperature probe. In our simulation environment, we need to adjust both of these temperatures in order to test how our application responds.

In order to control these temperatures, we use a **Numeric Adjuster,** which allows us to input and adjust the values we wish.

The first adjuster you are going to connect is an adjuster placed on the pin level drawing that will simulate the setpoint temperature that is input to the application. To do this, you will need to place a digital numeric adjuster on the Setpoint pin:



1 **Click the Setpoint pin on the pin level drawing.**

2 **Click** **.**

3 **Click where you want the Numeric Adjuster to appear. You may decrease the size of the adjuster by right-clicking on it, and selecting *Decrease size*.**

4 **Enter the temperature value 0.**

At the pin level, temperature values are expressed in Volts, rather than in °C. A simple conversion formula links the two quantities:.

$$\frac{T_{(°C)} \cdot 10}{255} \cdot 5V = T_{(V)}$$

Therefore, 0 °C = 0 V.

5 **Now, connect a numeric adjuster to the Temperature pin in the same manner, also setting the value to 0.**

A list of temperatures in °C and their corresponding values in Volts is shown in Table 4.

Table 4 A list of temperature values in °C and their corresponding values in Volts.

Setpoint Temperature (Setpoint pin)		Actual Temperature (Temperature pin)	
°C	**V**	**°C**	**V**
0	0	0	0
18	3.53	12	2.35
18	3.53	15	2.94
18	3.53	17	3.33
18	3.53	20	3.92

Connecting a Probe

In addition to adjusters, which fix input values, we also need to connect probes, to view output values. In this application, we will use all three types of probe to evaluate the response of our application:

During our simulation, it would be helpful to know whether the heat pump is running or not. Recall that we included a Pilot LED that would turn on when the heating came on. By connecting a **Numeric Probe** (which lets you view the value of the output in the Binary, Decimal, Hexadecimal and Octal number bases) to the HeatingIsON pin, we can view whether the LED is ON (voltage = 5.0 V) or OFF (voltage = 0.0 V).

To connect the numeric probe, proceed as follows:

1 Select the output pin called HeatingIsON.

2 Click **.**

3 Click where you want the Numeric Probe to appear (i.e. close to HeatingIsON).

In addition to seeing if the heat pump is running or not, we would also like to see its power rating. In our application, recall that the pump's power is directly proportional to the (positive) temperature differential between the actual temperature and the setpoint temperature.

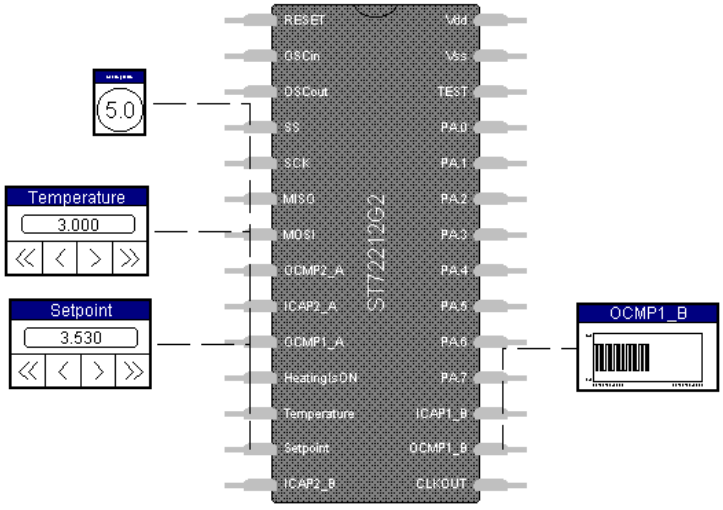
To view the input power to the heat pump, we can attach an **Oscilloscope Probe**, which allows you view the value of the output as a graph.

1 Select the Timer B output pin (called OCMP1_B).

2 Select the Oscilloscope Probe by clicking **on the toolbar.**

3 Place the probe where you wish it to appear.

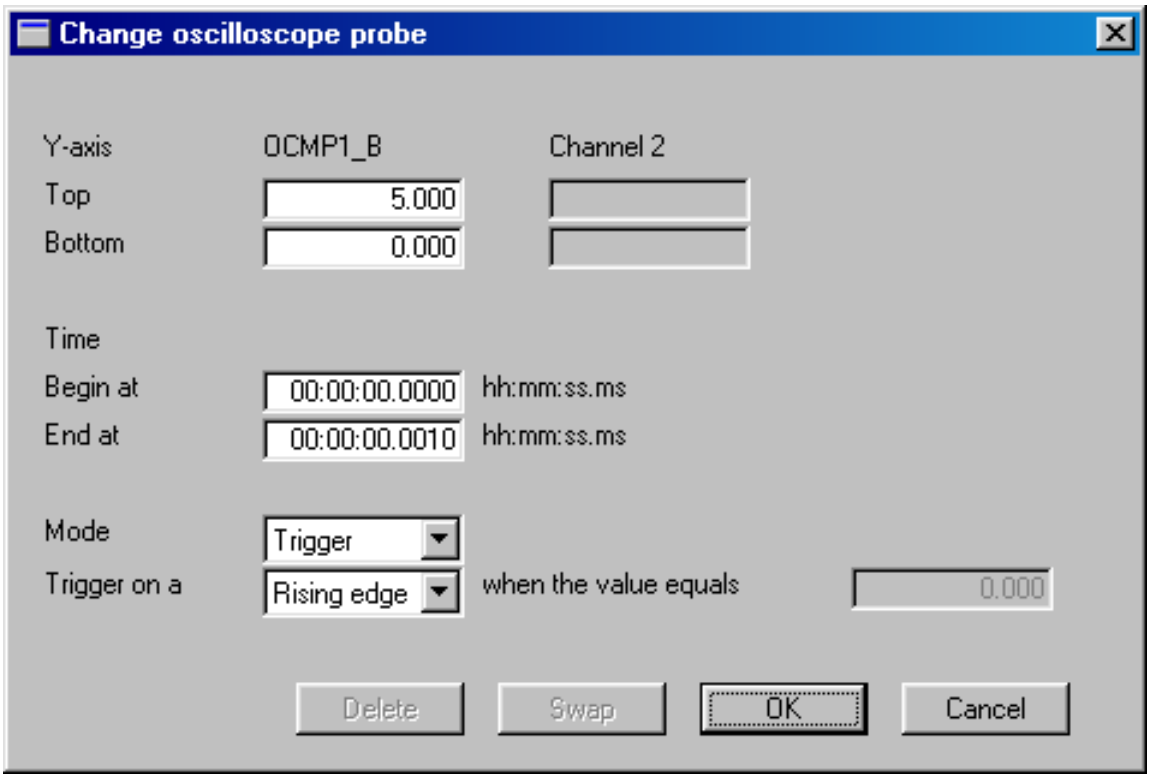
Your pin level diagram should now look something like the diagram at right.



Once the Oscilloscope Probe is placed, you need to set the variable display range as 0 to 5 and the display time to 0.001 s, meaning that the Y-axis on the graph displayed in the oscillator probe will start at 0 V and end at 5 V, and the X-axis (Time) will start at 0 s and end at 0.001 s(1):

1 **Double click the Oscillator probe once it is placed.**

The Change Oscilloscope Probe dialog box opens:



2 **In the Y-axis Bottom field, enter `0`.**

3 **In the Y-axis Top field, enter `5`.**

4 **In the Begin at field, enter`00.00.00.0000`.**

5 **In the End after field, enter `00:00:00:0010`.**

6 **Select the Mode value *Trigger*.**

7 **Select Trigger on a *Rising edge*.**

8 **Click OK.**

Now go to the view of the root scheme by selecting **`heating.sch`** under the Window menu.

To see what the signal values are in the various connection wires, you should place **Numeric Probes** in the main scheme diagram.

1 For a complete description of oscilloscope parameters and how to control them, please refer to Section 10.3.9 on page 153.

Since the heart of the heating control application is the state machine, it is useful to check that each input condition has the required effect on the state machine, and that the state machine takes the appropriate actions. To see what the current active state is, you can place **State Machine Probes** (probes that show the state machine's condition) in the main scheme diagram.

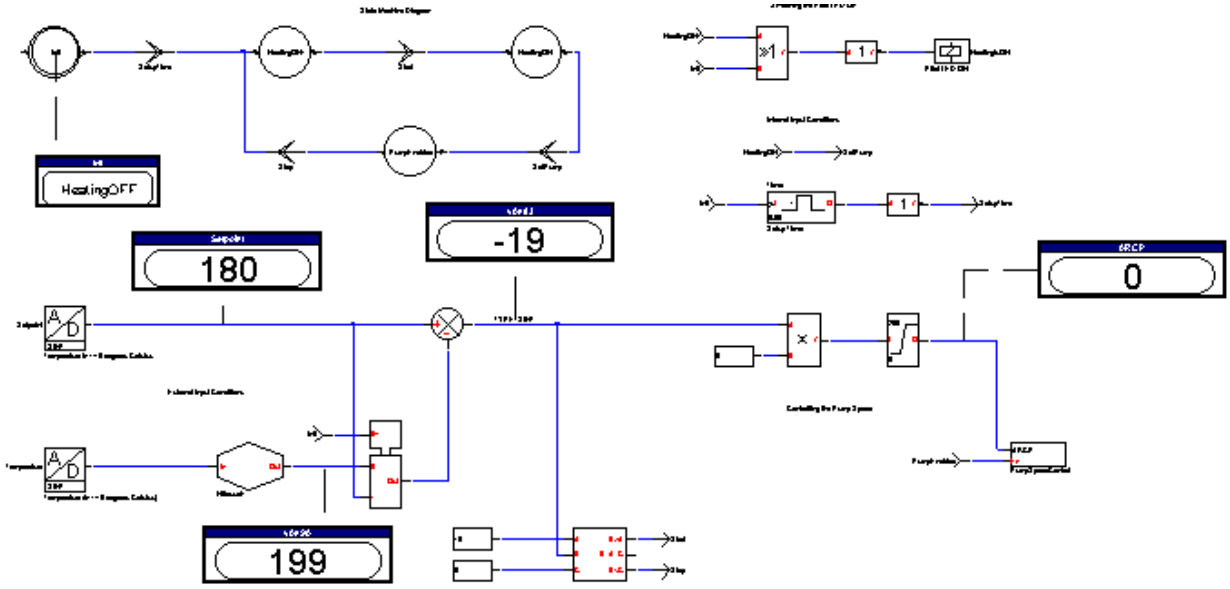
To probes in the root scheme:

1 **Click on the connection wire you wish. Note that State Machine Probes can only be connected to the initial state symbol in the State Machine Diagram.**

2 **Click** **to connect a Numeric Probe, or** **to connect the State Machine Probe.**

3 **Click where you want the Probe to appear.**

For example, you may wish to place **Numeric Probes** and a **State Machine Probe** in the main scheme diagram as follows:



You may also add additional probes to the subscheme **`filter.sch`** as desired, to better understand how the application is working.

Now you have placed your adjusters and probes, you can simulate your application, adjust the values input to it, and view the values generated by it.

3.5.19 Running the Simulator

You are now ready to run the simulator, and see how it reacts when you change the values input to it.

Click to run the simulation.

As you can see, the oscilloscope starts generating its signal. To simulate the behavior of the heating control system, change any value in the analog numeric adjusters placed after the adc pins. This causes the value of the pilot LED probe to be changed, and the shape of the oscillator signal to display discrete levels.

Try adjusting the input values of the Setpoint and Temperature pins by clicking the <, >, <<, >>, buttons on the analog numeric adjusters. The < and > buttons decrease and increase the values by 1 respectively, and the << and >> buttons decrease and increase the values by 10 respectively. Table 4, on page 49, shows a list of suggested values designed to allow you to view how the application reacts by starting or stopping the heat pump, depending on the difference between the Setpoint and Actual temperatures.(1)

When you want to stop running the simulator, click . To run again the simulator click . Note that before re-running the simulator, you can initialize it, by clicking: .

3.6 Now it's Up to You!

You've now completed the tutorial. In this tutorial you learned the three major steps involved in creating your own application using ST-Realizer:

- How to draw the application.
- How to analyse your scheme and compile your application code.
- How to simulate your application.

Furthermore, you learned what most of the ST-Realizer symbols do, and how and where to place them. You are now ready to start developing your own bug-free applications for the ST6 or ST7 families of microcontrollers using ST-Realizer. If you want to look at more examples, refer to Appendix B:"Sample Applications" on page 211.

1 Simulator time does not necessarily elapse at the same rate as real time (the time on your watch). The simulator time is generally slower than real time, depending on the hardware characteristics of the PC you are running ST-Analyser on. For this reason, in our application example (see Section 3.3.10 on page 35), we specified a periodic sampling every 0.01 seconds so that our simulation would not be too slow. If you find that the simulation takes too long to respond to changes in input values, you may need to change the periodic sampling time.

4 CREATING, OPENING AND SAVING PROJECTS

4.1 Project Files

Each application you design is stored in a **project**. It is recommended that each project have its own folder. ST-Realizer stores all schemes and subschemes associated with the application in the project folder.

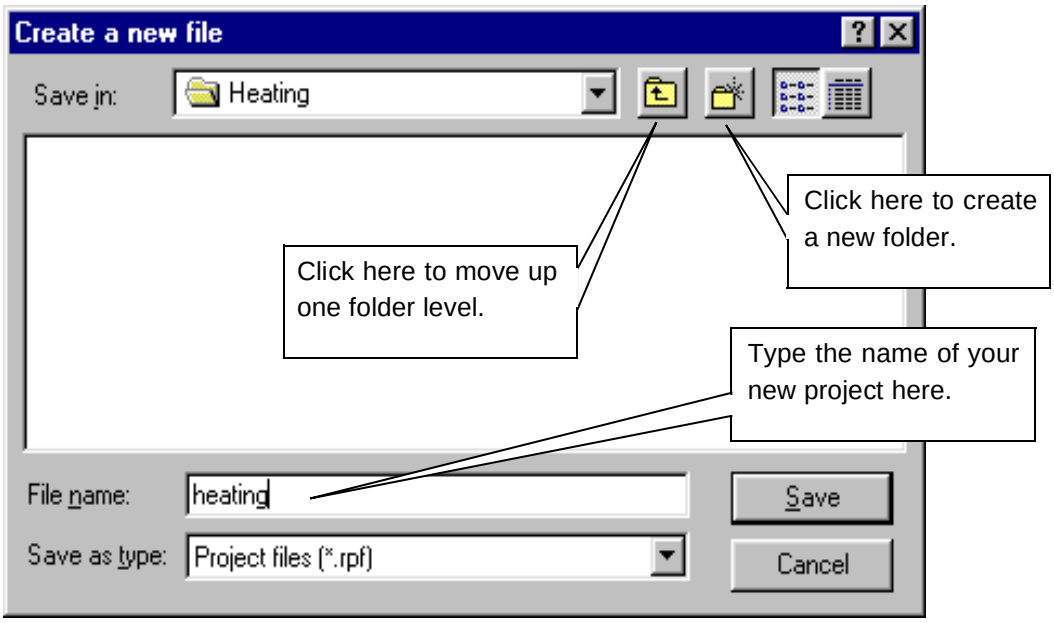
Once your have created and specified your project folder, ST-Realizer will create a ***project.rpf*** file, that contains project-specific path settings, the project's scheme names, target hardware information and compiler settings. The **project.rpf** file is in ASCII text format.

4.2 Creating a New Project

To create a new project:

1 Click *Project→New* in the cascading menu.

The ***Create a New File*** dialog box opens. If you haven't done so already, you may create a new folder for your project by clicking on the new folder icon shown.



2 Browse to the folder that you'll create your project in, and specify the name of the project file. An `.rpf` extension will be applied automatically. Click *Save*.

4.3 Opening an Existing Project

Note: Only one project may be open at a time.

To open a project:

1 **Click *Project→Open*. (Files with the `.rpf` extension are displayed automatically.)**

or,

Click *File→Open* or **. (You must specify the `.rpf` extension).**

The *Open a File* dialog box opens:

2 **Browse to the folder containing your project file, and either select it or type its name (`.rpf` extension) in the File name field.**

3 **Click *Open*.**

4.4 Opening Earlier Realizer Version Projects

The *Open a file* dialog box shown above can be used to open .ini project files from earlier versions of ST-Realizer. Both.ini projects and .rpf projects are fully compatible.

When opening projects from earlier versions, be aware that these projects still use the target hardware from these earlier versions. Simply reselect the target hardware (see Chapter 5 on page 59) once you have opened the earlier version project.

4.5 Closing a Project

To close a project:

Click *File→Close* or ***Project→Close.***

4.6 Saving Projects

Once you have defined your project, you should save it so that all of the configuration information you have updated is kept as a part of the .rpf file.

- If you try to close a modified project, you will be prompted to save it.
- Otherwise, to save a project, under the **File** menu, select ***Save***.
- If you wish to save your project to another filename (for example, create a back-up copy of the project), under the **File** menu, select ***Save As***, and type the name of the file that you wish to save the project to.

Tip: *If you save your project to a different folder, all of the schemes and subschemes associated with the project will also be saved to the new folder.*

5 SPECIFYING THE TARGET HARDWARE DEVICE

5.1 ST6 or ST7 Devices

Once you have created a new project, the next step is to define the ST6 or ST7 device type that the application will be loaded into. This will attach the hardware configuration of the ST6 or ST7 device (such as pinout and memory capacity) to the scheme that will describe your application.

This hardware data assures that the application is tailored to the target ST6 or ST7 device.

Note for ST6 users:

ST-Realizer does not support any RAM or ROM paging, except for static pages. Because of this, the ROM size is limited to 4 kilobytes and the RAM size to 128 bytes.

The *Max. ROM* and *Max. RAM* values determine the maximum size of the ST-Realizer application.

The I/O pins determine the number of input and output symbols that can be used in the ST-Realizer application.

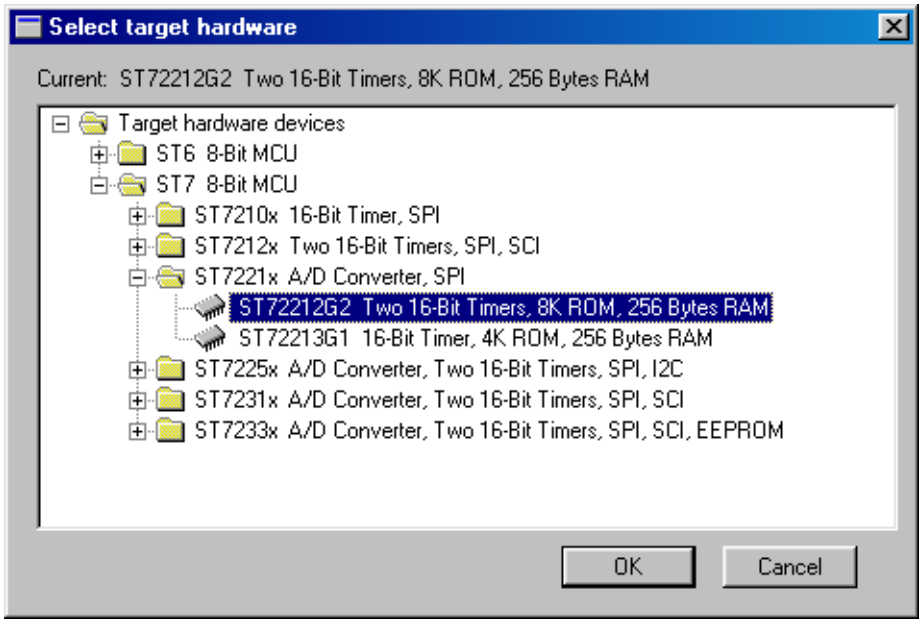
The EEPROM values determine how much EEPROM can be used by the symbols from the MAINPER.LIB symbol library. This symbol library contains symbols that use the EEPROM space to store their values.

5.2 Choosing a Target Microcontroller

5.2.1 Selecting the Target Microcontroller for a New Project

When you create a new project, just after creating your project.rpf file, you will be prompted to specify the target hardware for your project.

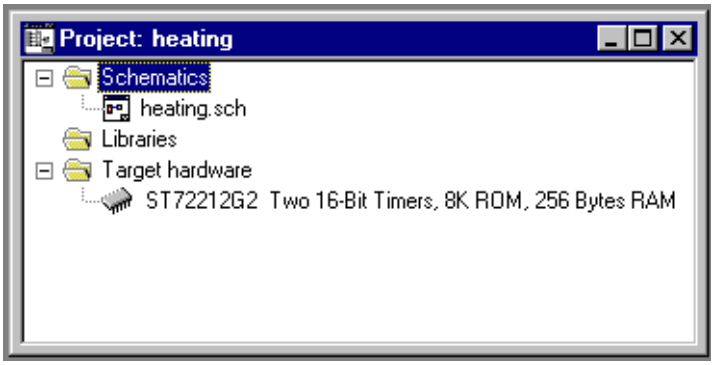
1 **Click the *Target Hardware* folder in the Select Target Hardware window. A browsable list of target hardware devices will appear in the window shown below.**


Select target hardware
Current: ST72212G2 Two 16-Bit Timers, 8K ROM, 256 Bytes RAM
Target hardware devices
ST6 8-Bit MCU
ST7 8-Bit MCU
ST7210x 16-Bit Timer, SPI
ST7212x Two 16-Bit Timers, SPI, SCI
ST7221x A/D Converter, SPI
ST72212G2 Two 16-Bit Timers, 8K ROM, 256 Bytes RAM
ST72213G1 16-Bit Timer, 4K ROM, 256 Bytes RAM
ST7225x A/D Converter, Two 16-Bit Timers, SPI, I2C
ST7231x A/D Converter, Two 16-Bit Timers, SPI, SCI
ST7233x A/D Converter, Two 16-Bit Timers, SPI, SCI, EEPROM
OK
Cancel

2 **Find the target ST microcontroller by clicking the device icons in the list until the name of the chosen device is displayed. Select the target device by clicking once on it.**

3 **Click OK to confirm.**
You may also double-click the line showing the device.

4 **Once you have selected your target hardware, the Project Viewer will open, similar to that shown at right:**



5.2.2 Changing the Target Microcontroller

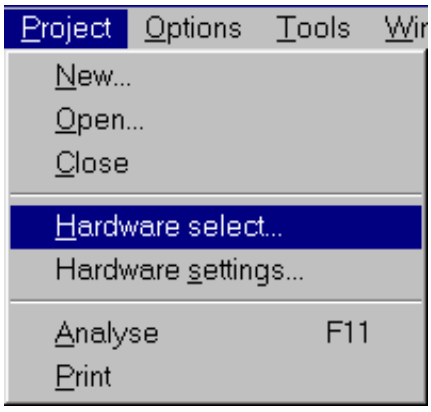
Once you have specified a target hardware device for a project, the configuration and a connection(1) information for that device is stored in the .rpf file for the project. If you change the target hardware device of a previously existing project, the new target hardware device information is also added to the .rpf, without losing the previously specified device information.

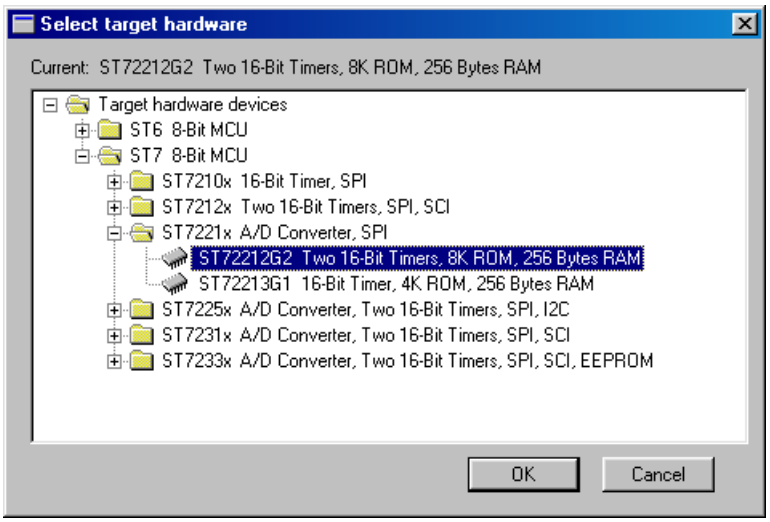
To change the target hardware device for an existing project, follow these steps:

1 **Open the Project whose target microcontroller you wish to modify.**

2 **Click *Hardware Select...* on the *Project* menu, or double-click the target hardware device in the Project Viewer.**

3 **The Select target hardware dialog box will open, showing the target hardware device previously selected. Simply browse to the new target hardware device desired. Click once to select it.**




Select target hardware
Current: ST72212G2 Two 16-Bit Timers, 8K ROM, 256 Bytes RAM
Target hardware devices
ST6 8-Bit MCU
ST7 8-Bit MCU
ST7210x 16-Bit Timer, SPI
ST7212x Two 16-Bit Timers, SPI, SCI
ST7221x A/D Converter, SPI
ST72212G2 Two 16-Bit Timers, 8K ROM, 256 Bytes RAM
ST72213G1 16-Bit Timer, 4K ROM, 256 Bytes RAM
ST7225x A/D Converter, Two 16-Bit Timers, SPI, I2C
ST7231x A/D Converter, Two 16-Bit Timers, SPI, SCI
ST7233x A/D Converter, Two 16-Bit Timers, SPI, SCI, EEPROM
OK
Cancel

4 **Click OK to confirm.**

You may also double-click the line showing the microcontroller. Note that the name of the current microcontroller is displayed on top of the dialog box.

1. If you add a new target hardware device to a pre-existing project, you may have to update the hardware connections by reconnecting I/O symbols to the appropriate hardware ports and peripherals. For more information, refer to “Connecting Input/Output Symbols to Microcontroller Pins, Ports and Peripherals” on page 80.

5.3 Hardware Configuration

5.3.3 Accessing Hardware Settings Dialog Boxes

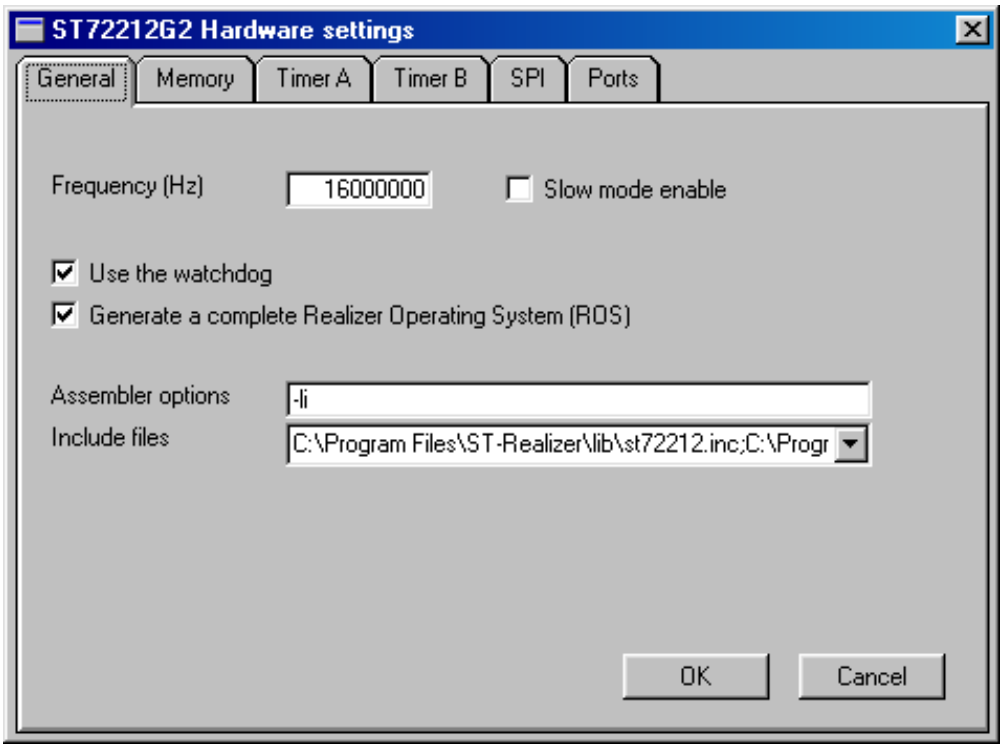
Follow these steps:

1 Click *Hardware Settings.* on the *Project* menu.

The *Hardware Settings* dialog box for the specified target microcontroller opens.

This dialog box shows a number of tabs that direct you to the following hardware configuration daughter dialog boxes:

- General tab, for configuring general hardware options.
- Memory tab, for setting hardware memory options.
- A tab corresponding to each of the target microcontroller's on-chip peripherals, allowing peripheral settings to be customized. For example, the ST72212G2 microcontroller hardware settings dialog box, shown below, has four peripheral setting tabs corresponding to each of its four on-chip peripherals: Timer A, Timer B, SPI and Ports.



2 Click the appropriate tab.

Note: There are two circumstances in which you may want to modify the settings in these windows: (a) when you build a new scheme, and (b) when you want to re-compile an existing scheme in order to customize how the program will operate when loaded into the microcontroller. In the first case you specify new settings, (or keep default settings); in the second case, you modify existing settings.

5.3.4 General Hardware Configuration

This is the first tab in the dialog box.

Note: Option bytes are not supported.

From the **General** tab, you can specify:

1 The oscillator frequency of the microcontroller.

Note: This frequency is the external frequency. The ST7 uses a internal frequency which is half the external frequency ($F_{in} = F_{Ext}/2$).

2 Whether or not the Watchdog function is enabled.

The Watchdog function is a peripheral included on each ST6 and ST7 microcontroller. Enabling the Watchdog initializes it and instructs ST-Realizer to refresh the Watchdog regularly. For more information on the Watchdog function, please refer to your microcontroller's datasheet.

3 Whether or not you want the ROS to be disabled.

The ***Realizer Operating System*** (***ROS***) is made up of macros or pieces of code that perform background tasks that must be added to an ST-Realizer application for it to be complete and ready to load into an ST device. In a certain sense, the ROS is similar the BIOS of your PC—ROS macros encompass such operations as chip initialization, I/O initialization, timer initialization and data memory initialization, that are essential to the running of your application.

Note: Refer to Chapter 2, "Introduction and Concepts" on page 3, for an overview of the running of an ST-Realizer program.

Alternatively, you can disable the inclusion of the standard ROS macros, and generate your own code to perform the ROS tasks instead. The tasks which you must provide macros for are described below.

To disable the ROS:

- **Click the Complete ROS check box. ROS is disabled when the check box is empty (unchecked).**
- **Click OK.**

If you disable the ROS, you must use an external program to perform the following functions:

- Call the following subroutines that are created by ST-Realizer:

PortInit, which initializes the I/O ports according to the ST-Realizer application.
RamInit, which initializes the RAM allocated by the ST-Realizer application.
RealInit, which initializes the ST-Realizer application.
RealMain, which executes the ST-Realizer application.

- The ***PortInit***, ***RamInit*** and ***RealInit*** must be executed once and the ***RealMain*** subroutine must be executed continuously.
- Perform all the interrupt management.
- Fill in the input variables that are used by ST-Realizer application and copy the output variables of the ST-Realizer application to the data registers of the I/O ports.

The input variables are:
`Apxd`
where **`x`** is the port name **`A`**, **`B`** or **`C`**, and **`d`** is the pin number **`0..7`**. This is a set of variables generated as the result of the A/D conversion. These variables are already allocated with the size of one byte.
`RTICK`
This is the one-byte timer tick variable. This must be filled with the number of ticks during the last execution of the RealMain subroutine.

The output variables are of the form:
`BUDRx`
where **`x`** is the port name **`A`**, **`B`** or **`C`** and each variable is one byte in length. Their contents must be copied to the data registers of the appropriate port.

4 Include files for user-defined macros.

If the scheme you are analysing and compiling includes a symbol that you created yourself, you must include the macro or macros linked to the symbol before analysing the scheme. For details on how to create your own symbols see “Creating Your Own Symbol” on page 165.

To include the macros linked to user-defined symbols:

i In the Include files field, enter the name and full path to the macro files linked to your symbols. To include more than one macro file, separate each path and file name with a semi-colon (;).

ii Click OK.

5 Assembler options.

Normally, you will not need to alter the default assembler options.

5.3.5 Memory Configuration

The second ***Hardware Settings*** dialog box tab is entitled **Memory**. In this tab, shown below, you can configure the target microcontroller's memory configuration options, such as:

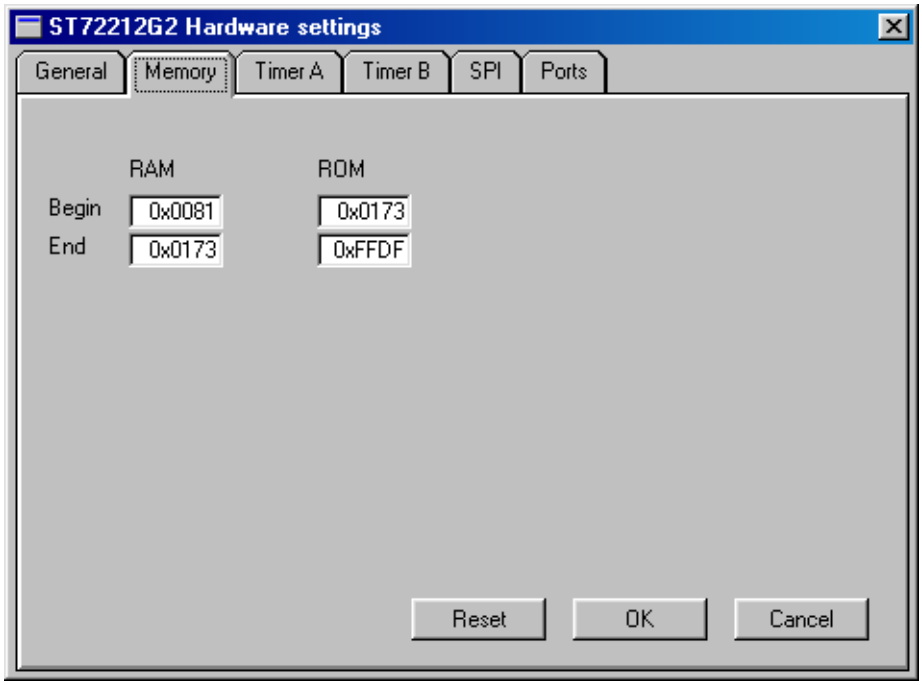
- The start and end of ROM (corresponding to the beginning of your application code).
- The start and end of RAM (corresponding to the beginning of your application variables).
- The start and end of EEPROM (if this memory type is included in your target microcontroller).

To set or update one of these parameters:

1 **Type a new value or overtype its current value.**

2 **Click OK.**

To reset the values to standard device settings, click ***Reset***.



Notes:

1. The default values found in these fields do not correspond to those reported in the microcontroller's datasheet because ST-Realizer reserves a portion of the available RAM memory for its own use. For example, for ST7 devices, ST-Realizer uses 12 bytes of RAM memory.
2. If the values in these fields are changed, there will be an impact on the application's variables and on the generation of the code.

5.3.6 Enabling Peripherals

Each peripheral belonging to the target microcontroller has an associated tab in the Hardware settings window. If you wish to use one or more of the peripherals in your application, you must enable the peripheral, which allows it to be initialized and configured. To do this:

1 **Click the appropriate tab to open the dialog box that describes the peripheral to be configured.**

The peripherals available vary depending on the target microcontroller device.

2 **Check the *Enable* box at the top of the peripheral's dialog box.**

The peripheral is now enabled and may be configured (using the rest of the options in the dialog box) as you wish.

For details on the peripherals available, and their configuration options, refer to the Microcontroller data book. See also Chapter 8, “The Main Symbol Library” on page 103.

6 CREATING, OPENING, SAVING SCHEMES

6.1 Schemes

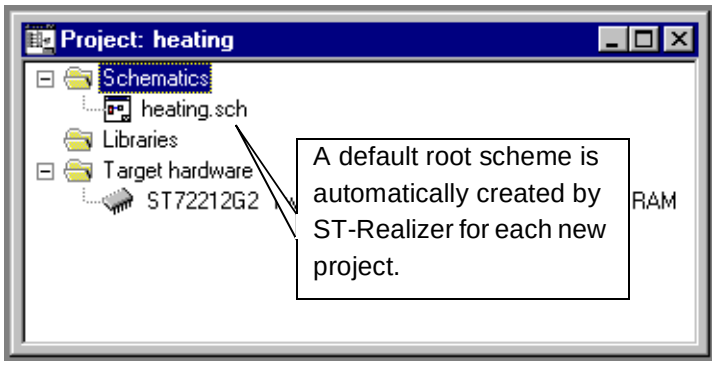
Once you have created or opened the project on which you want to work, you must create or open a scheme. A scheme is the sheet on which you design your application. When you have finished working on a scheme, remember to save your work.

You can export schemes in the Windows metafile (**.WMF**) format, so that you can import them into drawing or word processing packages.

This section describes how to create a new scheme, open an existing scheme, and save your work.

6.2 Creating a New Scheme

When you create a new project, an empty root scheme is created by default, taking the same name as the project. For example, if you created a new project called “Heating.rpf”, the root scheme called heating.sch will be created by default.



Tip: *If you wish to change the root scheme, right-click on the scheme in the schematics folder, and select* ***Change root schematic.*** *You will then be prompted to select (by browsing) the new scheme which will become the new root scheme for the project.*

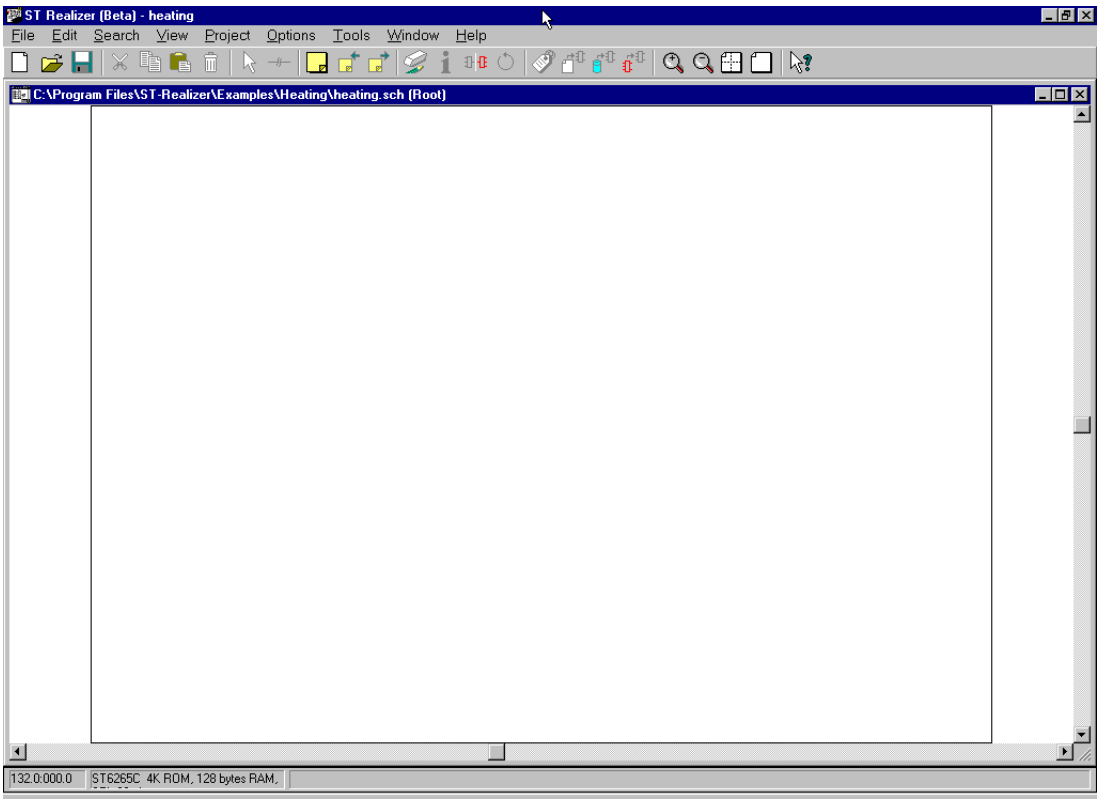
6.2.1 Opening the Root Scheme

The ***root (or “main”)*** **scheme** is the sheet on which you design the main part of your application and is created by ST-Realizer when you create a new project.

To open the root scheme:

- In the **Project Viewer** (shown above), double-click the root scheme (for example, **`heating.sch`**, as shown above) located under the *Schematics* folder.

A new blank worksheet opens where you can draw the root scheme of the application:


ST Realizer (Beta) - heating
File Edit Search View Project Options Tools Window Help
C:\Program Files\ST-Realizer\Examples\Heating\heating.sch (Root)
132.0:000.0
ST6265C 4K ROM, 128 bytes RAM,

6.2.2 Creating Subschemes and other Schemes

1 Click **.**

You may also click the *File→New→Scheme*.
The *Create a New File* dialog box opens.

2 Browse to your project folder and specify the name of the scheme file (.sch extension).

3 Click *Save*.
A new, blank scheme opens.

Tip:

*A new subscheme will not appear under the **Schematics** list in the **Project Viewer** until it has been connected to the project and the project has been **Analysed**. To connect a subscheme to a project, there must be a symbol in somewhere in the root scheme or in another subscheme which is connected to the new subscheme (refer to "Connecting a subscheme to a symbol in the root scheme" on page 92). If the new scheme is not a subscheme, it can only be connected to the project if it is declared as the root scheme.*

6.3 Opening a Scheme

1 **Click** **.**
You may also click *File→Open*.

The *Open a File* dialog box opens.

2 **Enter the path of your project, and specify the scheme name (.sch extension).**

3 **Click *Open*.**

Tip: *To open a scheme you've used recently, click its name at the bottom of the **File** menu.*

6.4 Saving Schemes

To save the scheme that is currently active, click .

To save all the schemes that are open, click ***Save all*** in the **File** menu.

To save a scheme under a new name,:

1 **Click *Save as* in the *File* menu.**

2 **Specify the new name of the scheme file (.sch extension).**

3 **Click *Save*.**

Note: You can also set up ST-Realizer to save your work automatically at a specified interval. See "Automatically Saving Your Work and Setting Screen Preference." on page 194 for further details.

Tip: *Subschemes will not be appear in the Project Viewer until you have performed an Analyse of the project. A saved .sch scheme file will be placed in the root directory of the project, but you will have to open it manually if does not appear in the Project Viewer.*

7 BUILDING SCHEMES

7.1 Schemes and their Components

A scheme is a collection of symbols connected to one another via lines named “wires”. Each symbol has input and output pins that allow them to be wired to other symbols. The symbols, pins and wires in a scheme can be assigned attributes to precisely define their configuration and behavior.

Schemes are at the heart of ST-Realizer. By designing schemes, you are in fact creating your application code.

In each application you create, you must have a **root (or “main”) scheme**. This is the scheme which contains the main body of your application—the core of the program which controls the sequential running of the application.

In addition to the root scheme, most applications also include subschemes that represent specific processing in your application. Subschemes usually contain those actions which must be performed in addition (and sometimes conditionally) to the actions in the root scheme. In particular, subschemes may be connected to events, such as timer interrupts, periodic events, hardware (external) input changes and peripheral operation. Subschemes can also be used to mask more complex operations, so that they do not “clutter” the root scheme.

7.2 Symbols

Symbols are the basic building blocks used to create an application with ST-Realizer. In essence, each symbol is a graphical representation of a portion of assembler code, usually representing a function or a short subroutine. Symbols can represent many coded entities such as: mathematical, logical, conversion and power management functions, constants, tables, subschemes/hierarchical sheets, states, input devices, output devices and sequential, counted or time-related events. The main symbol library in ST-Realizer (see Chapter 8 on page 103) encompasses all of the main functions required in assembler code, and should you need a very specialized symbol for your application, there is a symbol editor function that allows you to modify an existing symbol, or create an entirely new symbol, and save it to the library.

Most symbols in ST-Realizer’s main library have attributes which you must specify. These attributes allow you to specify many parameters, such as:

- Giving the symbol an application-specific name in order to identify it elsewhere in the scheme.
- Assigning the input or output data type.
- Specifying a time period in a time-related symbol.

The above list is by no means exhaustive. The modifiable attributes depend on the type of symbol. In Chapter 8 on page 103, you will find a complete list of all of the symbols in the ST-Realizer main library, complete with the modifiable attribute values available for each one.

Earlier, we mentioned that each symbol is in fact a graphical representation of a portion of assembler code called a macro. Later, in Chapter 11 on page 165, we will take a closer look at customizing symbols using the Symbol Editor, and at the assembler macros that define each symbol.

The remainder of this chapter will concentrate on how to manipulate symbols and their attributes, and how to wire together symbols to create an application scheme.

Note: With ST-Realizer, object naming is case-sensitive. In addition, spaces are interpreted as characters. Ensure that all object names are used consistently, otherwise, errors will result when you compile the application.

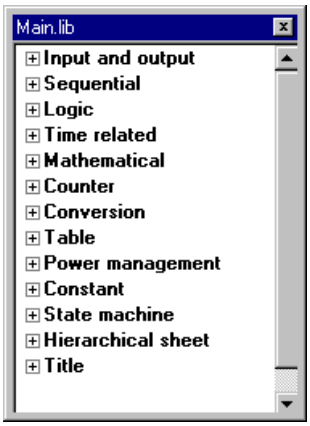
7.2.1 Placing and Controlling Symbols

Placing a symbol in the scheme

Symbols can be placed in a scheme in two ways:

1 Insert a new symbol (one that *does not already exist in the open scheme)* by choosing it from the **main library**:

i Click .



The ***main.lib*** (and ***mainper.lib*** if it exists for the target microcontroller) dialog box(es) open. The symbols are, by default, ordered by functional category (Hierarchical View). However, you can view the library of symbols alphabetically by right-clicking in the library dialog box and selecting ***Alphabetical View***.

You may also open any library via the **File→Open** menu sequence, by specifying the appropriate **`<filename>.lib`** file. ST-Realizer has with a number of symbol libraries in the **`<root_folder>\Lib`** directory, including a some that are microcontroller-specific.

Tip:

The most commonly-used symbols are grouped in two libraries:
***main.lib**, for symbols that use values to be stored in RAM, and **mainper.lib**, for symbols that use values to be stored in permanent, non-volatile storage.*
*When you build a scheme, you use a set of symbols that are copied from these libraries. The list of the symbols actually selected constitutes the **local library** attached to the scheme.*
***Note** that when you copy a scheme, you also copy the attached local library.*

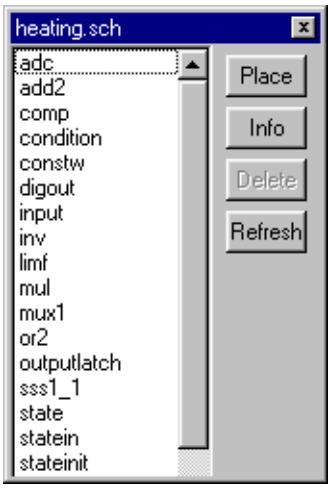
ii Double-click the name of the symbol you wish to place or select the symbol by clicking once on it, right-click the mouse without moving it from the library dialog box, and select *Place*

iii A square box will appear next to the cursor, indicating the size and position of the symbol you have chosen. Move the cursor to where you want to place the symbol, then click once.

2 Make a copy of a symbol that *already exists in the open scheme* (symbol from the **local library**):

i Click .

A dialog box (having the name of the open scheme in the title) opens. It contains a list of all the symbols used in that scheme.



ii Select the symbol you want to place. Click **Place**.
You may also double-click the name in the list.

iii Drag and drop the ghost box associated with the symbol down to the new location of the symbol.

You can obtain information about the symbol by clicking **Info**, prior to placing it.

Tip:

The local library attached to a scheme can be saved for further use. For example, to enrich the set of symbols available for another scheme.
*To do this, click the **Save as** option in the **File** menu, and specify a **.lib** file type, keeping the original name of the scheme.*

Selecting a Symbol, Wire or Group of Objects

As with any drawing package, before you can modify an object or a group of objects you must first select them.

To select items, you must be in selection mode (the cursor is in the form of an arrow). This is the default mode. However, if you need to activate

selection mode:

- **Click** **.**

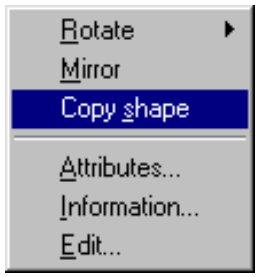
An object is selected when it is outlined by a red box. You may select one or more objects as follows:

- **Select one object by clicking it.**
  Take care not to click a symbol attribute as this will open the dialog box for the attribute.
- **Select a group of objects by dragging a box around them.**
  Put the cursor at one corner of the area you wish to select. Keeping the left mouse button pressed, move the cursor diagonally across the area you want to select until the whole area is outlined by a red box.
  When you release the mouse, you will see that each individual object found in the area you outlined has been selected.
- **Select multiple objects by pressing SHIFT while simultaneously clicking each object one by one.**
  Each item you select is surrounded by a red box, indicating that it is selected.
- **To select all segments of a connected wire, double-click one segment of the wire.**

Copying a Symbol

You can copy a symbol in two ways:.

1 Right-click the symbol to be copied. A popup menu will open. Select ***Copy shape***.
Drag and drop the ghost box of the symbol where you want to place it. You will be prompted to specify new attribute values for the symbol.



Tip:

The choices shown on the popup menu can vary depending on the type of symbol. For example, for Constant type symbols, a ***Value*** *option is also displayed, or for symbols connected to microcontroller output pins, such as* ***digout****, a* ***Connect*** *option will be shown.*

2 Select the symbol to be copied.
Click .
Drag and drop the ghost box of the symbol where you want to place it. Note that attribute values are also copied.

To copy a symbol to the clipboard, click .

Copying a Group of Symbols

You may copy either a group of symbols or a scheme (portion or entirety). Note that you will also copy all other objects in the group, such as wires.

1 Select the group of symbols to be copied.

2 Click in the toolbar.

3 Place the ghost box where you wish to place the copied objects.

To copy a group of symbols to the clipboard, select the group then click with the right mouse button.

The pop-up menu shown at right will appear. Select **Copy**.

To copy the selection to the scheme page choose **Duplicate**.

Pasting an Object from the Clipboard

Click , then drag and drop the ghost box down to the new location of the symbol.

Moving a Symbol or Group of Objects

To move a symbol or a group of objects:

- **Select the symbol or group of objects.**
- **Place the cursor on the selected group and drag and drop the ghost box with the four-headed arrow pointer to the new location of the symbol.**

Note that the wire connections attached to a symbol are moved with the symbol.

Deleting a Symbol or Group of Symbols

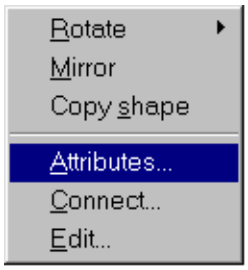
Select the symbol or the group of symbols. Click or press the **Del** key.

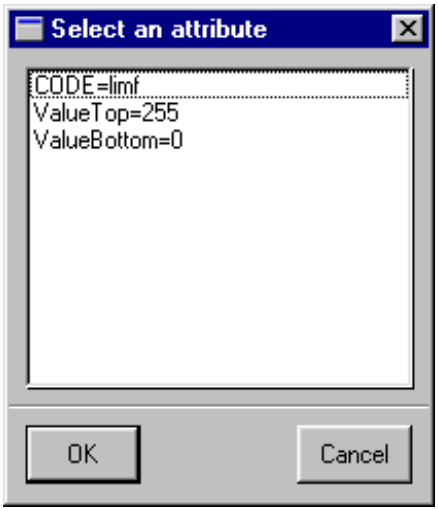
Note that you delete also all other objects in the group, such as wires. To delete a symbol and place it in the clipboard, select the symbol and click .

Changing a Symbol's Attributes

Symbols can have a variety of attributes that depend on the symbol type. When you place a symbol on a scheme from the symbol library, you are prompted to specify these attributes. However, they can be changed at any point. To do this:

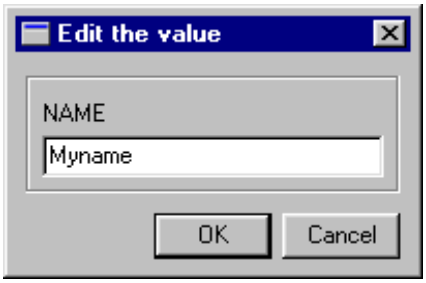
- **Right-click the symbol.**
  A popup menu opens.





•Click Attributes.
The ***Select an Attribute*** dialog box opens.

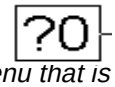
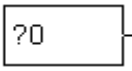
•Double click the line of the attribute you want to change.
An *Edit the Value* dialog box opens.



The field label in this dialog box (NAME, in the example) depends on the value to be changed for the specified attribute.

- **Enter the new value, then click OK.**

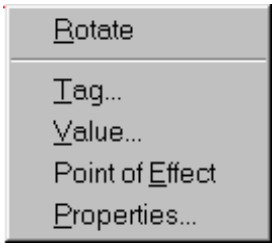
Tip:

To change the value of the Constant symbols  *or*  *, you may also click the* ***Value*** *entry in the popup menu that is specific to this type of symbol. The same* ***Edit the value*** *dialog box opens.*

Changing the Symbol Attribute Preference Settings

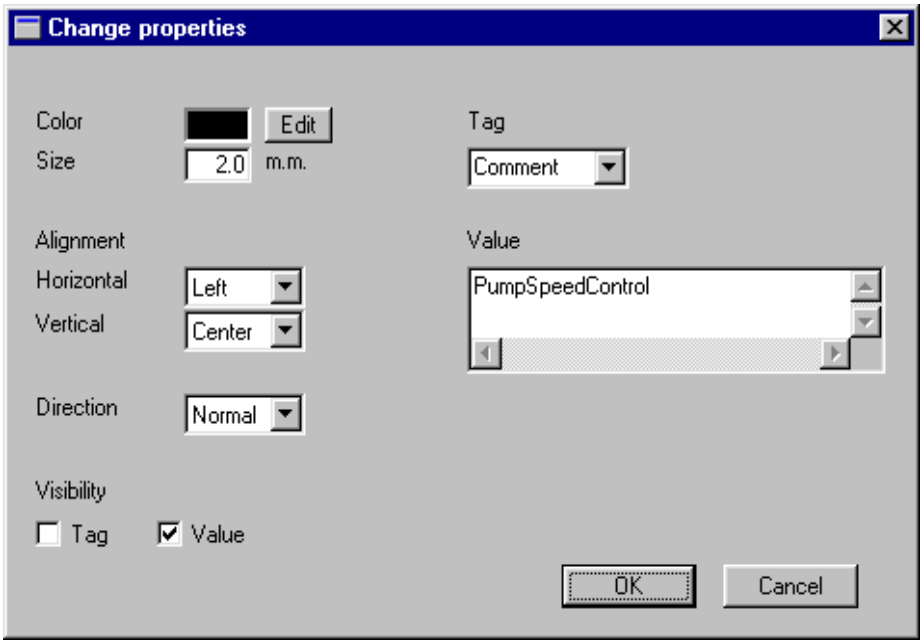
You may also change the manner in which an attribute is displayed on a symbol:

- **Right-click the attribute field in the symbol of interest.**
  A popup menu opens.



- **Select *Properties...***

The following dialog box opens:



- **Change the preference settings to the desired values.**
  Select the appropriate **Alignment** and **Direction** settings in the corresponding drop-down lists.
  To change the color of the text, click **Edit** and select the new color from the displayed palette.
  To have tag name and value displayed on the scheme check the appropriate box.
- **Click OK to confirm the changes you have made in the preference settings.**

Mirroring a Symbol or Group of Symbols

Select the symbol(s), then click .

You may also use the ***Mirror*** option in the popup menu associated with the symbol (right-click the symbol).

To mirror a group of symbols, select the group then click with the right mouse button.

A popup menu displays. Choose **Mirror**.

Rotating a Symbol or Group of Symbols

Select the symbol(s) you wish to rotate, then click .

The selected symbol will be rotated by 90° counterclockwise.

You may also use the ***Rotate*** option in the popup menu associated with

the symbol (clicking the symbol with the right mouse button).

To rotate a group of symbols, select the group then click with the right mouse button.

A popup menu displays. Choose **Rotate**.

Viewing Symbol Information

Select the symbol about which you want to view information.

Click .

An information box opens.

When you have finished viewing the symbol information, click **OK**.

7.2.2 Wiring Symbols Together and Connecting Application Inputs/Outputs

Almost all symbols have at least one input and one output pin to which you connect wires (the only exceptions are some subscheme symbols). Wiring two symbols together creates the data flow between them. By default, the input pin(s) are to the left of the symbol and the output pin(s) to the right.

This section describes how to wire symbols together, control the attributes of wires and how to connect external application inputs and outputs to the appropriate target microcontroller ports or peripherals.

Drawing Wires between Symbols

To wire two symbols together:

- **Select wiring mode by clicking** **.**
  The cursor changes to a crosshair, indicating that it is in wiring mode.
- **Place the cursor on the pin of the first symbol, where you want the wire to start.**
  The crosshair snaps onto the pin when it comes into snapping distance. An x indicates the point to which the crosshair is snapped.
- **Click when the crosshair is snapped to the pin of the first symbol, where you wish the wire to start.**
  ST-Realizer will now draw a wire that follows the cursor.

Tips:

If you want to define your own wire corners, click twice where you want each corner to be.

- **Move the cursor to the pin of the second symbol, where you**

wish the wire to end.

- **Click when the crosshair is snapped onto the appropriate point.**
  The two places where you clicked are now connected by a wire.
- **Right-click the mouse or press the ESC key or click** **to finish wiring.**
  The two symbols are now connected by a wire.

Using Automatic Wiring:

You may also let ST-Realizer draw wires for you.

Automatic wiring simplifies the task of wiring symbols together by:

- Automatically choosing the shortest path between the two symbols to be connected (Auto wiring), and creating corners where required.
- Automatically rerouting wires when a symbol is moved (Auto rerouting).

Both these options are enabled by default.

For details see “Wire Drawing Options” on page 201

Copying a Wire

Select the wire to be copied.

Click .

Drag and drop the ghost box of the wire where you want to place it. Note that attribute values are also copied.

To copy a wire to the clipboard, click .

Pasting a Wire from the Clipboard

Click , then drag and drop the ghost box down to the new location of the wire.

Moving a Wire

Select the wire, click once on it, and drag and drop the ghost box with the four-headed arrow pointer down to the new location of the wire.

Deleting a Wire

Select the wire or the group of wires. Click or press the **Del** key.

Note that you delete also all other objects in the group, such as symbols.

To delete a wire and place it in the clipboard, select the symbol and click .

Mirroring a Wire

Select the wire, then click .

Rotating a Wire

Select the wire, then click .

The selected wire is rotated by 90° counterclockwise.

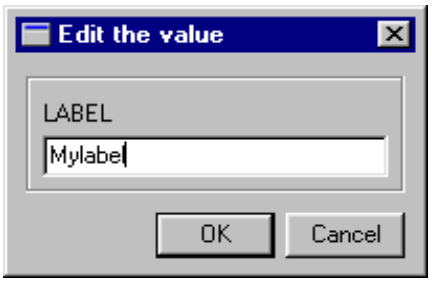
Changing a Wire's Attributes

Click the wire with the right mouse button.

A popup menu opens:

Click the name of the attribute you want to change.

An ***Edit the Value*** dialog box opens:



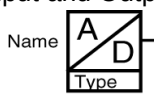
Enter the new value, then click **OK.**

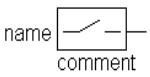
Connecting Input/ Output Symbols to Microcontroller Pins, Ports and Peripherals

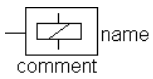
Application input and output symbols must be connected to the ST6 or ST7 microcontroller input and output pins, ports or peripheral control registers in order for the application to function.

Note:

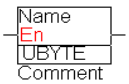
Application Input and Output symbols are:

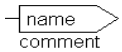
adc: 

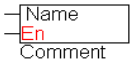
digin: 

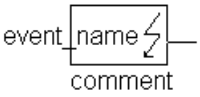
digout: 

input:

inputlatch: 

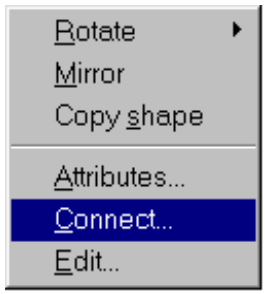
output: 

outputlatch: 

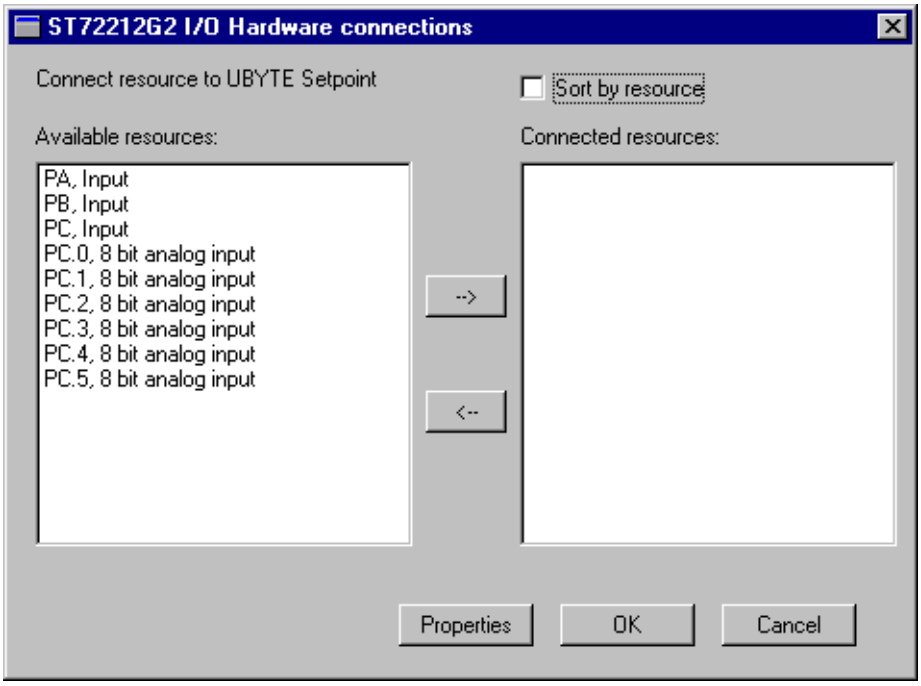
event 

eventenable

To connect these symbols to a target microcontroller port or peripheral:



- **Right-click or double-click the I/O symbol.**
  This popup menu opens:
- **Click *Connect.***
  The *I/O Hardware Connections* window for the target microcontroller opens. The resources available in this window vary depending on the target microcontroller.
- **Select the appropriate device pin.**



- **Click the right arrow or double click the selected device pin name. Click OK.**

Tips:

*To have connections for the device sorted in pin order, check the **Sort by resource** box.*
*If the peripheral register or bit that you wish to connect does not appear in the list at the left side of the window, click on the **Properties** button. This will open the **Hardware settings** dialog box, and allow you to enable the peripheral that you desire to use. When you return to the **Hardware connections** dialog box, the registers or bits of the peripheral you just enabled should now be included in the list.*

7.3 Working in Schemes

This section describes general functions and utilities available when you are working in a scheme, such as:

- How to use the viewing options.
- Viewing hidden attributes.
- How to print schemes.
- How to add a title or comment to a scheme.

Using the Zoom-In View

Click , then click the blank area in the scheme worksheet.

You may also click the **Zoom in** entry in the **View** cascading menu you obtain by clicking the blank area in the scheme worksheet.

Note: Zooming in magnifies the view by 200%.

Zooming in on a Selected Area

Click , then select the area of the scheme you want to zoom in on.

Using the Zoom-Out

Click , then click the blank area in the scheme worksheet.

You may also click the **Zoom out** entry in the **View** cascading menu you obtain by clicking the blank area in the scheme worksheet.

Note: Zooming out reduces the magnification by 50%.

Using the Full Screen View

Click .

You may also click the **Full view** entry in the **View** cascading menu you obtain by right-clicking any blank area in the scheme worksheet.

Redefining the View Center of your Scheme

Click , then click where you want the new centre to be on the scheme.

You may also click the **Pan** entry in the **View** cascading menu you obtain by clicking the blank area in the scheme worksheet.

Printing the Currently Active Scheme

Click the **File-Print** sequence in the main menu bar. Continue the normal printing dialog and click **OK** when ready.

- **Set Up the Printer**

  On the File menu, click **Printer setup**.

  The standard Windows Print Setup dialog box opens. Refer to your Windows documentation for further information.

  Click **OK** when you have finished setting up your printer.

- **Choose a Printer Font**

  On the **File** menu, click **Printer setup**, next click **Properties** and the **Font** tab.

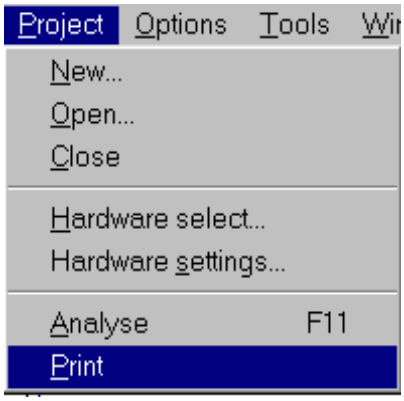
Tip:

For best results, use the True-Type fonts that come standard with the your Windows environment.

Printing all Project Schemes

Click **Project** and select **Print** in the drop-down menu:

Continue the normal printing dialog and click **OK** when ready.

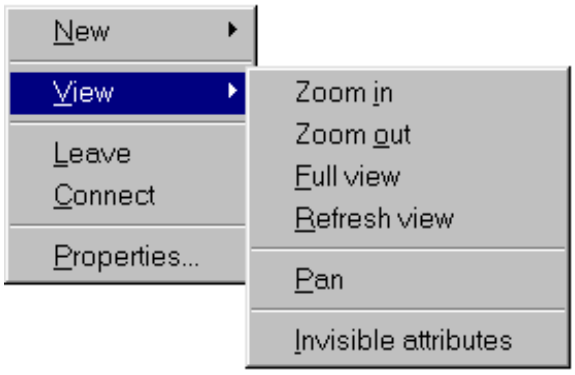


Viewing Hidden Attributes

When the shapes and pins that make up a symbol are created, attributes can be assigned to them to define additional characteristics.

These attributes are not visible by default when you design an application, since they only refer to parts of the symbol itself. ST-Realizer enables you to view hidden attributes:

Click the **Invisible attributes** entry in the **View** cascading menu



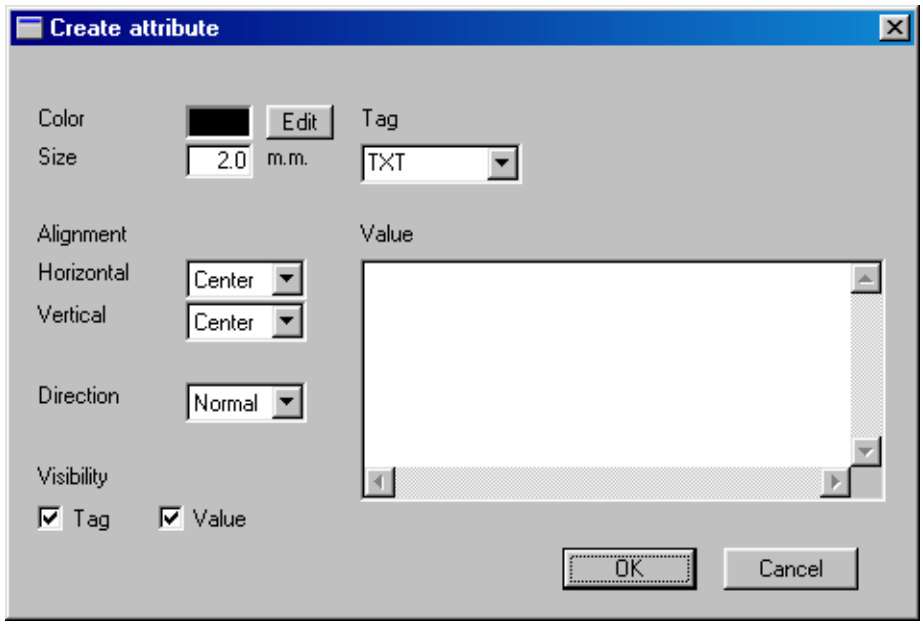
you obtain by clicking the blank area in the scheme worksheet.

When the **Invisible attributes** option is selected, all attributes are visible.

Placing a Title in the Scheme

1 Right-click any blank area in the scheme.
A popup menu appears.

2 Select the ***New/Attribute*** options.
The **Create attribute** dialog box opens.



3 Specify the following options:
- **TAG = TXT**
- **Enter the title text in the Value edit box.**
- **Under Visibility, there are two options. If you wish only the title text that you typed to appear, select Value only. If you wish the attribute tag to appear as well (in this case "TXT = *title text*"), select both Tag and Value.**

4 You may specify color, size or alignment values for the title.

5 Place the heading by dragging it where you want and click **OK.**

7.4 Subschemes, Execution Conditions and Events

Subschemes are exactly what their name implies—additional schemes that are subservient to the root scheme. Their appearance is similar to the root scheme, in that they also contain symbols and wires. However, subschemes exist apart from the root scheme, and are only executed when called upon via a subscheme symbol in the root scheme.

As previously discussed, there are three reasons to create a subscheme:

- To include complex portions of the main loop, thus saving space in the root scheme and making it easier to reuse processes. In this case, the subscheme is executed as if it were a part of the main loop.
- To include parts of the application that are event-driven. (Events can never be placed in the root scheme.) Subschemes can be assigned either a single execution condition, which will apply to the entire subscheme, or alternatively, can include any number of event symbols. More will be said about execution conditions and events shortly.
- To save functional parts of your application (analogous to subroutines) that you may wish to reuse in other applications. Subschemes are saved in their own files (**.sch** files) and can be easily copied to other ST-Realizer projects and reused. You may also save customized subschemes symbols to a library, to be accessible by all projects. (Subscheme symbols are described below).

Designing a subscheme is no different than designing an ordinary scheme, with one exception—a subscheme has connections to its root scheme via a ***subscheme symbol***. The subscheme symbols are named **sssp_q**, where **p** indicates the number of inputs you need for your symbol and **q** the number of outputs. For example, **sss2_1** is a subscheme with two inputs and one output.

When you want to use a subscheme, you must therefore first think about its connections: what inputs does the subscheme need to deliver its output. Once you know this, you can choose the correct subscheme symbol from the main library.

Subschemes not linked to events (either by definition of execution conditions or the inclusion of event symbols) are basically annexes to the root scheme, and therefore, the same rules that apply to root schemes must apply to them.

However, those subschemes to which execution conditions are attached, or in which event symbols are embedded, are special cases, and the rest of this section is dedicated to describing how to configure subschemes with execution conditions or event symbols.

7.4.3 Description of ST-Realizer Events

Events are a general concept (described in Section 2.6 on page 8), and allow for the conditional execution of code within an application. As outlined above, if an event is applied to

an entire subscheme, it is called an execution condition; if it is applied to only a portion of code within a subscheme, the event takes the form of an event symbol.

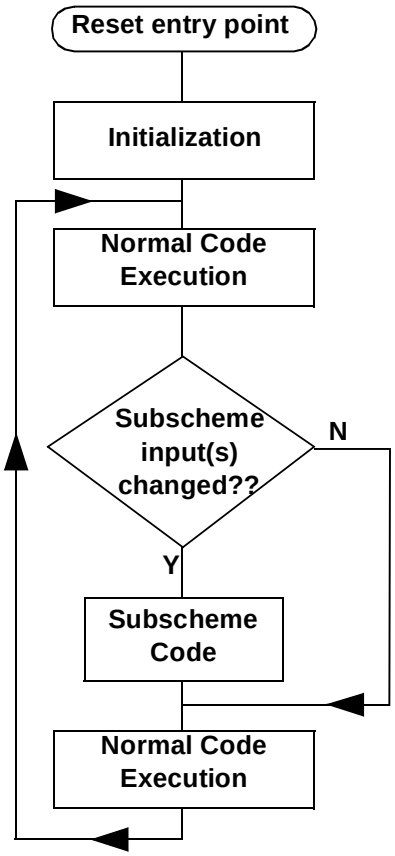
The following paragraphs give a list of the principle types of events—those that are hardware-independent, meaning that they are available on all microcontrollers—and those that are hardware-dependent, meaning that they make use of peripherals that are only available on certain microcontrollers.

Events that are independent of the target hardware device:

- **Upon subscheme input change**

  Code contained in a subscheme is executed each time one of the input signals coming from the parent scheme has changed. This is similar to an If..Else switch in standard programming algorithms.

  The diagram at right shows a schematic flow diagram of an application with a subscheme having this type of event (either as an execution condition or an event symbol).

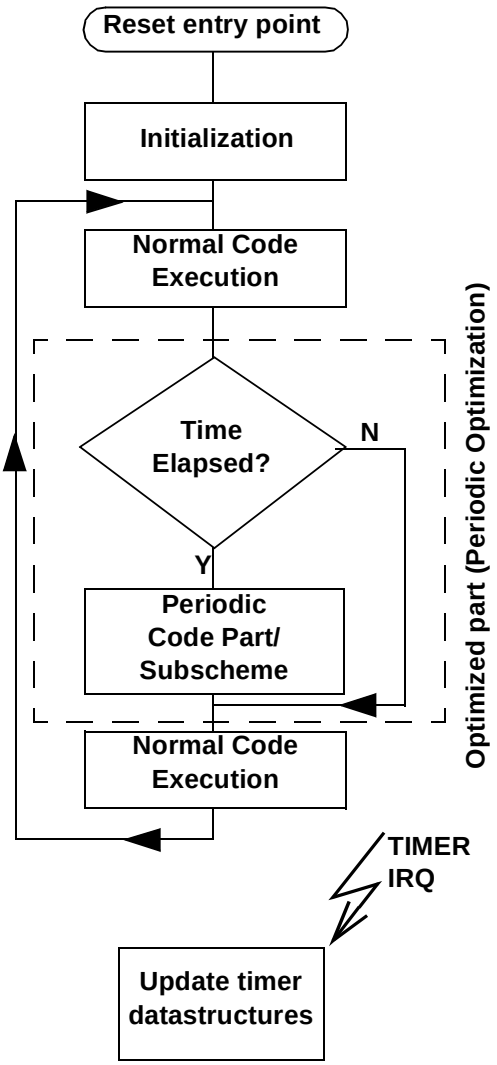


- **Periodic Events**

  A periodic event, or execution condition, acts at the level of the main loop. Inclusion of a periodic event causes a counter based on the number of base clock timer ticks elapsed in previous main loop cycles to measure a certain period of time. (Therefore, the period specified must always be a whole number multiple of the timer tick's value).

  Once the period of time specified has elapsed, the subscheme or periodic code part will be executed and the periodic counter will be reset.

  A periodic event or execution condition is analogous to an If..Else statement, where the If condition is whether a specified number of ticks have been counted *at the beginning of each main loop cycle*.



  For example, at the beginning of each main loop cycle, the number of timer ticks elapsed during the previous main loop cycle is written to a variable called "rtick"(1). In other words, during the execution of main loop cycle *n*, the value of *rtick* is equal to the number of ticks elapsed in main loop cycle *n-1*. Because periodic events and execution conditions add together *rtick* values to count elapsed time, they are relatively imprecise timing methods and should not be used when the application requires a very precisely timed event to take place.

  For example, imagine that the you wish to define an event period equal to 5 timer ticks. Say that during the first main loop cycle, the number of timer ticks elapsed is equal to 2. The value of *rtick* for the duration of the second main loop cycle will therefore be equal to 2. However, during the second main loop cycle, because of other events or interrupts, the execution time is longer, and 4 timer ticks elapse. The value of *rtick* will be 4 for the

1. A more detailed look at how the variable "rtick" is related to the timer tick is provided on page 131.

duration of the 3rd main loop cycle, so that when the periodic event is evaluated, it adds together the values of *rtick* that it has received for all completed main loop cycles, and arrives at a total of 2 + 4 = 6 timer ticks. Evaluation of elapsed time in periodic events is always performed using an “equal to or greater than” condition. Therefore, the event will take place during the 3rd main loop cycle. However, there will be an imprecision of at least 1 timer tick in this particular case, because the specified period was 5 timer ticks and the actual elapsed time was 6 timer ticks at the beginning of the 3rd main loop cycle.

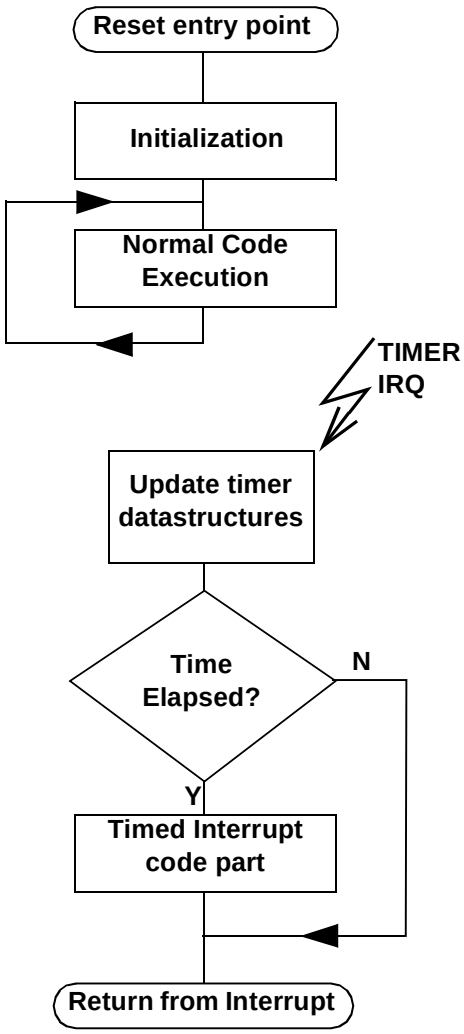
- **Timed interrupts**

  Timed interrupts are a very precise method of timing an event, because a timed interrupt is executed independently of the main loop of the application, and measures time directly from the base clock timer tick.

  While periodic events count the cumulative value of elapsed timer ticks at the beginning of each main loop cycle, timed interrupts count each elapsed timer tick as it occurs. This means that as soon as a specified amount of time (i.e. a specified number of timer ticks) has elapsed, the interrupt is immediately executed.

  In other words, a timed interrupt is triggered directly from the hardware clock (for example, Timer 1 or Timer A) measuring out the timer ticks, rather from the evaluation of main loop variables (such as *rtick*) which count the number of timer ticks elapsed in the previous main loop cycle.

  Therefore, no matter what the main loop is doing, when the specified time has elapsed, the main loop is interrupted and the timed interrupt code or subscheme is performed immediately.



Event that are hardware-dependent are:

- **Peripheral interrupts**

  These are interrupts that occur when a specified peripheral has signalled the application in some way.

  An example would be an SCI peripheral, which uses an RS232 protocol transmission. If you specify an SCI interrupt, the interrupt would occur (independent of the main loop) when the end of the transmission has occurred.

7.4.4 Execution Conditions

Execution conditions can be applied to subschemes, such that the subscheme is only executed when certain conditions are met—such as a timed interrupt, or upon a subscheme input change.

Subschemes with execution conditions allow you to perform tasks in addition to the main loop, if the execution condition is met. In this way, subschemes with execution conditions are analogous to subroutine operations.

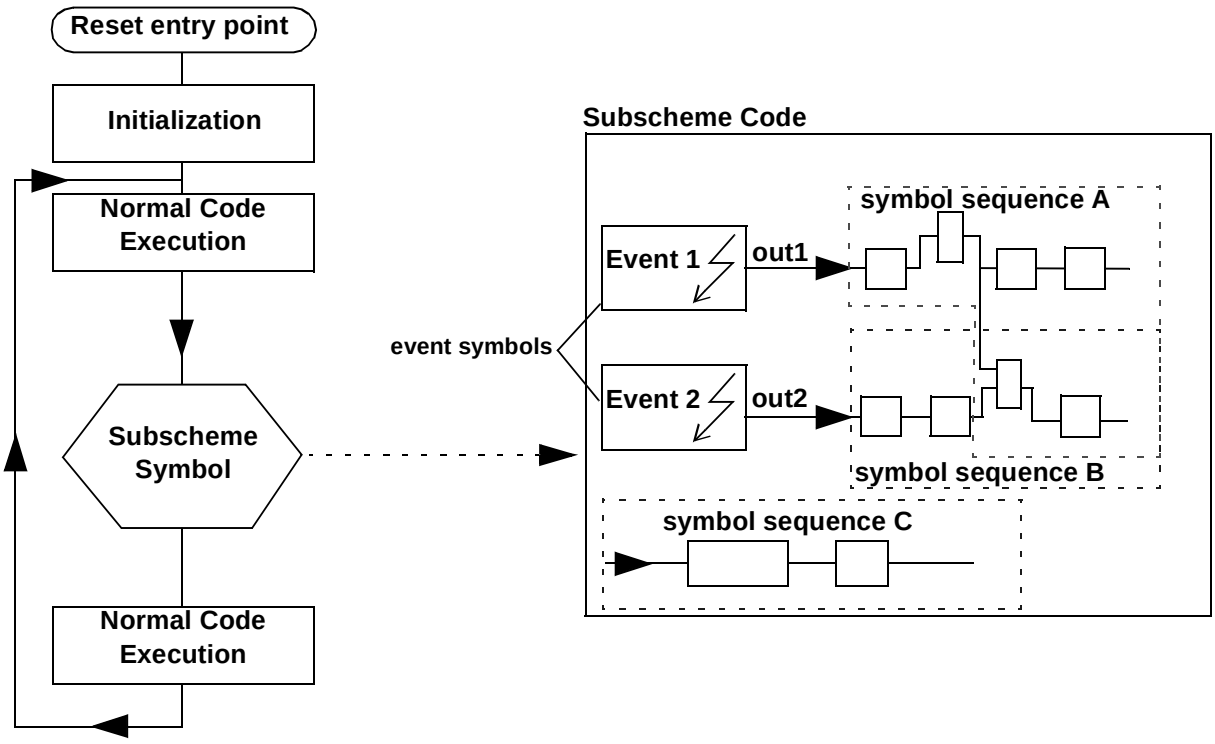
While a root scheme runs in a constant loop from the time the application starts to when it stops, subschemes are only performed when specified by the execution conditions.

There are a variety of execution conditions that one can assign to a subscheme, and these can vary depending on the target microcontroller for which the application is designed, and furthermore, upon which of the microcontroller's peripheral have been enabled for use by the application.

7.4.5 Event Symbols

Event symbols are analogous to execution conditions, but while execution conditions apply to an entire subscheme, an event symbol is placed within a scheme, and acts to trigger only certain portions of the subscheme. When the conditions necessary to trigger an event symbol are met, the event symbol outputs a binary value of "1". When the event symbol's conditions are not met (i.e. **the event is not triggered**), the event symbol outputs a binary value of "0". However, if none of the event symbols in a subscheme are triggered, the subscheme is effectively invisible to the rest of the program and no code within it is executed.

Event symbols are assigned in much the same way as execution conditions—all of the same options are available. However, because more than one event symbol can be included in a single subscheme, they can be used as a means of imposing a range of conditions on the same, or similar sequences of code. Because of this functionality, event symbols were conceived more as a means of controlling hardware-dependent interrupts, than as an all-purpose condition trigger.



If we revisit the flow chart shown in Section 2.6.5, we can see how different event symbols can be incorporated in the same subscheme, and how they influence how (and under which conditions) the code in the subscheme is executed.

In this schematic example, we have placed a subscheme symbol in the midst of our normal code loop (i.e. the root scheme) that references a subscheme containing two event symbols.

If neither of the conditions defined by the event symbols are met, the subscheme is ignored. However, if either or both of the events are triggered (meaning that their conditions are met), then the code in the subscheme executed in the following manner:

- If Event 1 is triggered (by the event's condition being met):
  - Symbol sequence A will be performed with output signal equal to "1" (out = 1). In other words, the symbol sequentially following the Event 1 symbol will have the binary value "1" as its input signal, and the rest of code sequence A will be executed in accordance.
  - Symbol sequence B will be performed with output signal equal to "0" (out2 = 0). In other words, the symbol sequentially following the Event 2 symbol will have the binary value "0" as its input signal, and the rest of code sequence B will be executed in accordance.
  - Symbol sequence C will be performed unconditionally (there are no event symbols connected to this code sequence).

- If Event 2 is triggered:
  - Symbol sequence A will be performed with out1 = 0.
  - Symbol sequence B will be performed with out2 = 1.
  - Symbol sequence C will be performed unconditionally.
- If neither Event 1 nor Event 2 is triggered, **no part** of the subscheme code will be performed.

7.4.6 Compatibilities Between Types of Events and Certain Symbols

There are certain rules concerning which types of events can be placed together in the same subscheme, because some types of events have priority over others, and furthermore, some types of events are incompatible when placed together in the same subscheme.

Event types that may be placed together in the same subscheme:

- Any number of peripheral interrupts. When these interrupts occur simultaneously, the program will give priority to the peripheral in accordance with the IRQ settings.
- A timed interrupt and any number of peripheral interrupts. Once again, execution priority is given in accordance to IRQ settings.
- Periodic events and upon subscheme input change events may be placed together in the same subscheme. Execution priority has less importance, because neither of these events are independent of the main loop (i.e. each event is triggered by a value that is augmented incrementally with the base clock timer tick). If both events occur simultaneously, the event symbol closest to the upper left-hand corner of the subscheme will be treated first.

Event types that may never be placed together in the same subscheme:

- More than one timed interrupt.
- A periodic event and a timed interrupt.
- A periodic event and a peripheral interrupt.
- An “upon subscheme input change” event and a timed interrupt.
- An “upon subscheme input change” event and a peripheral interrupt.

Note: If any of the above event types are placed separate event symbols in the same subscheme, an error message will result when you run ST-Analyser.

Compatibilities between Events and Time-Related Symbols

- Periodic events and time-related symbols are completely compatible, because they both count elapsed time in an identical manner.

7.4.7 Subscheme Operations

In the remainder of this section, we will describe how to create a subscheme and assign execution conditions to it.

Creating a Subscheme

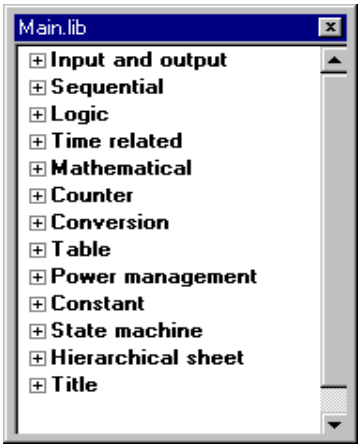
You can create your subscheme just a you would create a new scheme (refer to Section 6.2.2 on page 68). Once you have opened the new scheme window, design your subscheme with these guidelines in mind:

- Place the symbols that represent the function the subscheme is supposed to execute.
- Place as many **portin** and **portout** symbols as necessary. These symbols are used to establish the links (in and out) between the root scheme and the embedded subscheme.
- Make sure that the names for the **portin** and **portout** symbols are the same in the root scheme and in the subscheme.
- Input ports are normally to the left, and output ports to the right, of the symbol.
- Choose a name for your scheme and save it. A ***.sch*** file is created with the name you specified.

Connecting a subscheme to a symbol in the root scheme

Once your subscheme has been drawn:

- **Open the root scheme.**
- **Click** **.**



The *Main.lib* list dialog box opens. Select the sss-type symbol you want to place. You will select the sssp_q symbol that has p portin symbols and q portout symbols in the future subscheme.

- **Click Place.**

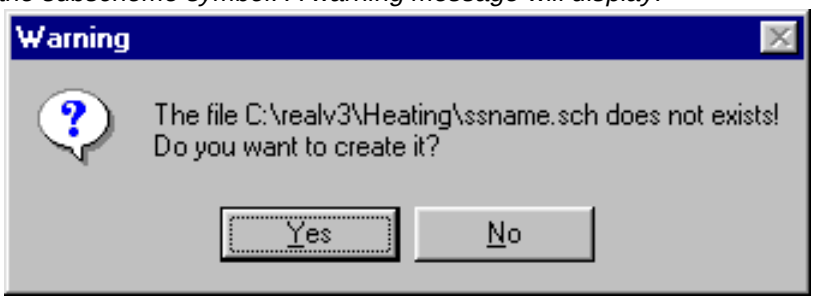
The sssp_q symbol will be placed in the root scheme. For example, the sss2_1 symbol is shown at left

- **When prompted to enter the “SCHEME” attribute, type the name of the subscheme (without the *.sch* extension).**

The subscheme is now inserted in the root scheme: double-clicking the symbol in the root scheme causes the subscheme to open.

Tip:

You may always add subscheme symbols to the root scheme even before you have created the subschemes themselves. For example, you could place the subscheme symbol, name it (using the same name you intend to give the subscheme file). Then, to create the subscheme, all you need to do is double-click the subscheme symbol. A warning message will display:



*Click **Yes**. A new scheme window will be opened and you can begin drawing your subscheme.*

Opening a Subscheme

To open a subscheme whilst in the root scheme:

- **Double-click its symbol in the scheme.**

To open a subscheme from the **Project Viewer**:

- **Click on the name of the subscheme under the *Schematics* folder.**

Assigning an Execution Condition to a Subscheme

To assign an execution condition to a subscheme, you must be in the subscheme.

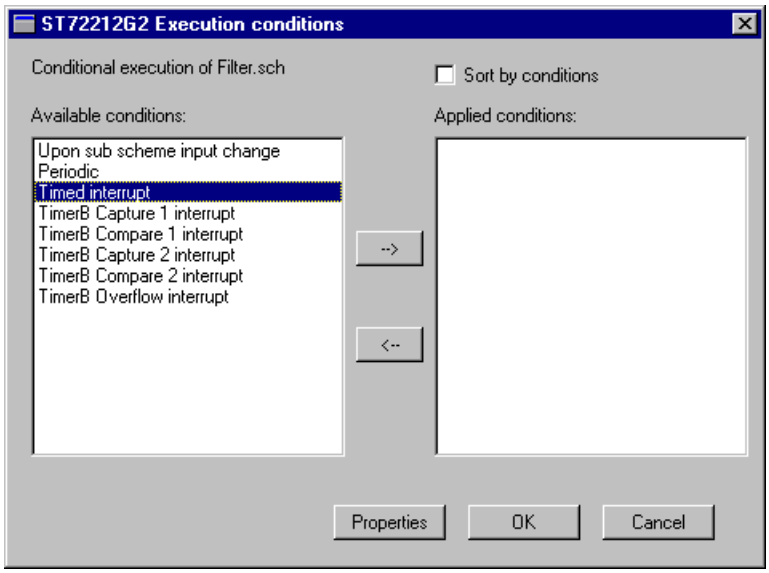
- **In any blank area of the subscheme, right-click the mouse to bring up the following popup menu:**

- **Select *Execution conditions*.**

The ***Execution conditions dialog box*** for the current target microcontroller opens (ST72212G2 in the example shown below).

It shows the list of events that can occur during the main loop execution and cause a specific action to be performed, namely the execution of the code generated by the subscheme.

This list depends on the target microcontroller and also which peripherals on the microcontroller have been enabled. For this reason, it is important to enable all of the peripherals that your application needs to function (see “Connecting Input/Output Symbols to Microcontroller Pins, Ports and Peripherals” on page 80) before you define the execution conditions in any subschemes.



In the example, the microcontroller ST72212G2 has had one of its peripherals, Timer B, enabled. The resulting choice of execution conditions are:

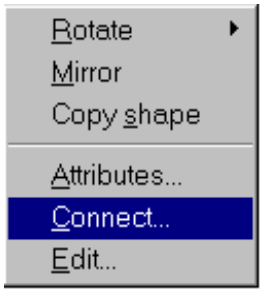
- Each time there is a subscheme input change.
- Periodically, by specifying a fixed time between each consecutive execution.
- According to a timed interrupt.
- According either one of five Timer B interrupts (these are only available because the Timer B peripheral has been enabled).

Placing and Configuring an Event Symbol in a Subscheme

By placing an Event symbol in the subscheme, the execution of the entire subscheme, or part of it, can be triggered by an event determined by a specific symbol within the subscheme.

- **Place the event symbol in the appropriate location in the subscheme.**
- **You will be prompted to provide values for the attributes Comment and Name.**
  The Name attribute will identify the event in the subscheme. In the example shown below, the value of the Name attribute is “My_Event”.
- **Wire the event symbol to the rest of the subscheme diagram.**

- **Right-click the symbol area.**
  A popup menu opens:

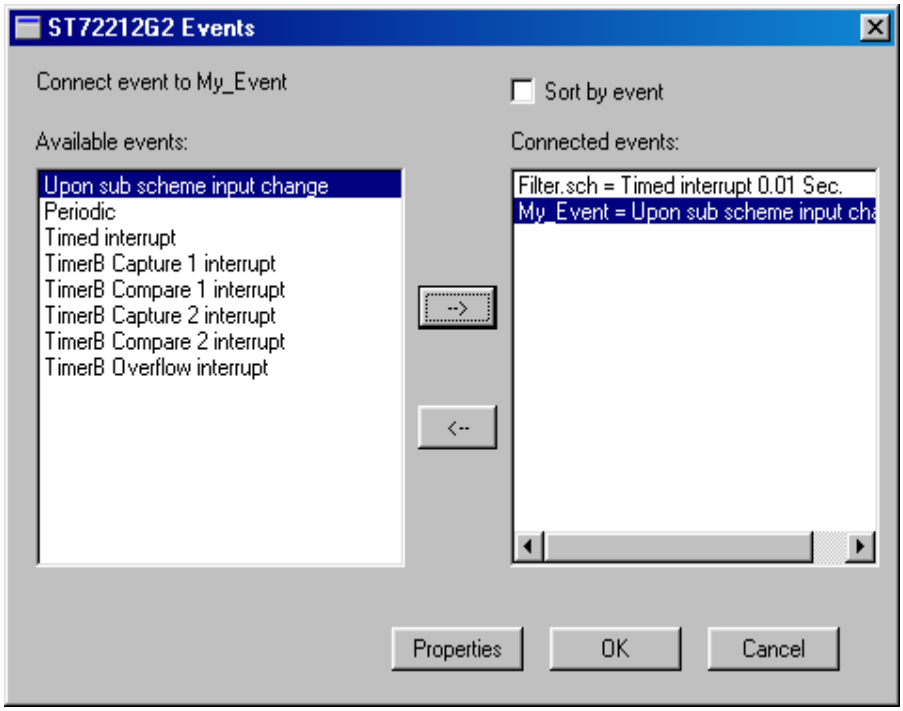


- **Select Connect.**

The ***Events dialog box*** for the current target microcontroller opens (ST72212G2 in the example).

It shows the list of events that can occur during the main loop execution and cause a specific action to be performed, namely the execution of the code in the subscheme that follows the event symbol.

This list depends on the target device.



In the example (ST72212G2), the event “Upon subscheme input change” has been associated with the event “My_Event”.

Configuring a Subscheme Input Change Event

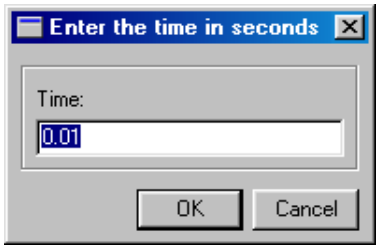
Select ***Upon subscheme input change*** in the left window pane and click the right arrow (or double-click the line).

Click OK.

Configuring a Periodic Event

Select ***Periodic*** in the left window pane and click the **right arrow** (or double-click the line).

A new dialog box opens, for you to specify the frequency of the code execution (for example, specify 0.01 s if you want the code to be executed once every one-hundredth of a second). Click **OK**.



Click **OK** in the events dialog box.

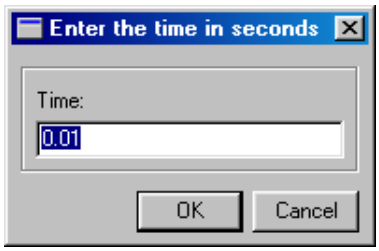
Note: Remember that the time specified in a periodic event should be a multiple of the base clock timer tick value. To change the value of a timer tick, refer to page 131.

Tip: *Don't use periodic events for time-sensitive subschemes where high precision is required. Because the execution of the code is not interrupt-driven, in some instances the precision of the timing may be unacceptable. Periodic events are best used to measure long periods of time where precision is less important.*

Configuring a Timed Interrupt Event

Select ***Timed Interrupt*** in the left window pane and click the **right arrow** (or double-click the line).

A new dialog box opens, for you to specify the time interval between two consecutive timer interrupts.



Click **OK**.

Click **OK** in the events dialog box.

Tip: *Timed interrupts are a very precise means of timing events and can be applied to short delays.*

Configuring an NMI Interrupt Event (ST6 devices only)

The execution of the code generated by the subscheme is triggered by the occurrence of a non-maskable software interrupt.

Select ***NMI Interrupt*** in the left window pane and click the **right arrow** (or double-click the line).

Click **OK**.

Configuring an Input Interrupt Event

The code generated by the subscheme is executed when an interrupt occurs on an input line.

Select the appropriate ***input Interrupt*** in the left window pane and click the **right arrow** (or double-click the line).

Click **OK**.

Configuring a Timer Overflow Event

The code generated by the subscheme is executed when a timer overflow occurs.

Select the appropriate line in the left window pane and click the **right arrow** (or double-click the line).

Click **OK**.

Configuring Other Peripheral-Dependent Events

The code generated by the subscheme is executed upon the occurrence of an event interrupt specific to the peripheral (for example, end of SPI data transfer, ARTimer Compare interrupt, etc.). Note that the peripheral must be enabled (refer to Section 5.3.6 on page 66) before these specific interrupts will appear in the Execution Conditions or Events dialog boxes.

Select the appropriate line in the left window pane and click the **right arrow** (or double-click the line).

Click **OK**.

Disconnecting a Subscheme from its Execution Conditions

If you wish to remove the execution conditions from a subscheme, perform the following:

- **In any blank area of the subscheme, right-click the mouse to bring up the following popup menu:**

- **Select *Execution Conditions*.**
  The ***Execution conditions dialog box*** for the current target microcontroller opens. It shows, at the right side of the window, a list of the applied conditions that have been attached to the subscheme.
  Note that more than one condition can have been specified.
- **Select the event to be removed.**
- **Click the left arrow.**
- **Click OK.**

Returning to the Root Scheme from a Subscheme:

Click the subscheme area out of any symbol with the right mouse button.

A popup menu opens:

Select **Leave**.

7.5 Table Symbols

There are two types of data tables that you can include in your scheme as a way of converting or treating data: lookup tables and index tables. Both of these table symbols are found in the main library, under the “Table” functional category.

Lookup tables convert input values to output values by matching their values in the table. If the input value is not found in the table, a default output value is assigned. An example of a lookup table is:

Input value	Output value
def	0
10	82
15	128
30	67
35	48

According to the above table, when the input value is 10, the table symbol will output the value 82. When the input value is not equal to either 10, 15, 30 or 35, the output value will be equal to 0.

Index tables are single column tables of output values only, however, each consecutive value has a different indice. For example, the first value in the table would correspond to the default value, the second value would have indice = “0”, the third to indice “1”, etc. This allows values to be output sequentially, such as would be the case if you had a loop where at each iteration, the output value changes. An example of an index table is:

Output value
0
45
96
132
145

According to the above example, if the input value was “2”, the index table symbol would output “132”.

Tables can store values in the Binary, Decimal, Hexadecimal and Octal formats.

The first line in a table defines the default output value (**`def`**). This is the value that is output if the input value is out of the range defined by the table.

Once you have placed a table, you can either import data from an ASCII text file or enter the data directly into the table. The following diagram shows the format of ASCII files—note that commas are used to separate table entries:

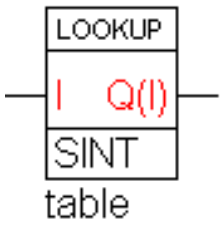
ASCII File Contents:	Table Values
aah 10h ,14h 20h ,30h 40h ,100h	

	input	output
def		aa
0	10	14
1	20	30
2	40	100

ST-Realizer stores table data in ASCII text files that have the extension **.TAB**.

Inserting a Table Symbol

The Index and Lookup table symbols are named ***indextable*** and ***lookuptable*** and are stored in the main library under the Table functional category.

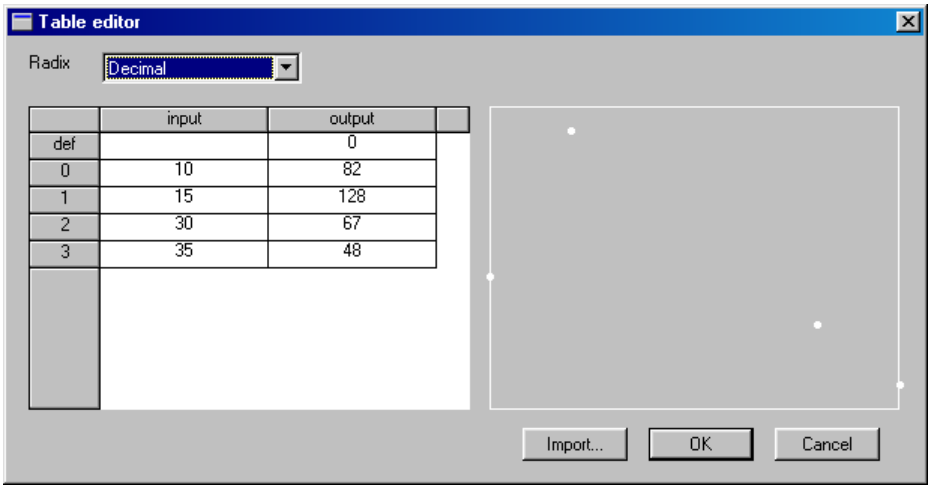


Changing table data format

Double-click inside the table symbol.
The ***Table editor dialog box*** opens.
Select the format you want to use from the drop-down list in the **Radix** combo box.
Click **OK**.

Editing table data

Double-click inside the table symbol.
The ***Table editor dialog box*** opens. (The figure below shows the lookup table used as an example on the previous page.)



Double-click the cell whose value you want to change.

To insert a new row, click where you want to insert the row then press the **Insert** key on your keyboard. For more rows, press **Insert** again.

Click **OK**.

Note that the Table editor dialog box shows an area for the graphical representation of the table. X-axis is the input (or index) axis, Y-axis is the output axis. The smallest Y value is on the bottom border, the highest Y value is on the top border, the first value is on the left border, the last value is on the right border.

Importing files Into tables

Double-click just next to the table.

The ***Table editor dialog box*** opens.

Click the **Import** button.

The ***Import file dialog box*** opens.

In the Directories box, double-click the folder holding the file you want to import.

Note: The ASCII file you wish to import must be in the format shown on the previous page—i.e. table entries must be separated by commas.

In the list below the ***File name:*** field, double-click the file you want to import.

Click **OK**.

8 THE MAIN SYMBOL LIBRARY

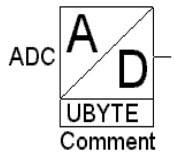
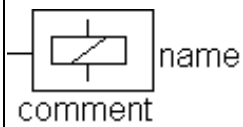
This chapter contains a listing of the symbols available in the main library (**main.lib**). Some of them may be also included in the “persistent” (non volatile) library (**mainper.lib**) (these are mentioned in the corresponding explanation column, where applicable).

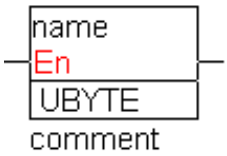
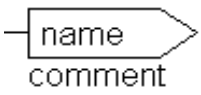
The main symbol library is divided into the following functional categories: Input and output, Sequential, Logic, Time related, Mathematical, Counter, Conversion, Table, Power management, Constant, State machine, Hierarchical sheet and Title.

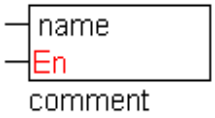
Each of these functional categories, and the symbols they contain are explained in the following sections.

8.1 Input and Output Symbols

These symbols provide input and output links between the application you create and the target microcontroller’s pins, ports and peripherals. It is important to remember to connect each of the symbols from this category to resources on the target hardware device. Refer to “Connecting Input/Output Symbols to Microcontroller Pins, Ports and Peripherals” on page 80 for instructions on how to do this.

Symbol	Symbol Name	Description
	**adc**	**Analog-to-Digital Converter Symbol.** The NAME attribute is used for connecting this symbol to a hardware port. For this, double-click the symbol. The TYPE attribute is used to define the variable type (UBYTE, SBYTE, UINT, SINT, LONG, or WORD). The COMMENT attribute is used in the report file.
	digin	**Digital Input Symbol.** The NAME attribute is used for connecting this symbol to a hardware port. For this, double-click the symbol. The COMMENT attribute is used in the report file. Type = BIT
	**digout**	**Digital Output Symbol.** The NAME attribute is used for connecting this symbol to a hardware port. For this, double-click the symbol. The COMMENT attribute is used in the report file. Type = BIT

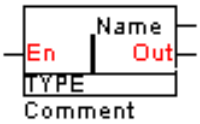
Symbol	Symbol Name	Description
	event	**Event Symbol.** Can be used only in a subscheme to cause the symbols that follow it in the subscheme code flow to be executed when the associated event occurs. Output type = EVENT
	eventenable	**Event With Enable Symbol**. Can be used only in subschemes to connect an event to the scheme it is in, thus causing an event-driven execution of the scheme. E type = Boolean. When TRUE it enables the execution of the event-driven subscheme. OutDigital representation of the event (type = Event). CountEvent name, the user is prompted for the symbolic name (Count). The connection is made to a micro controller event/interrupt and is invisible. Remarks:None
	input	**Input Symbol.** The NAME attribute is used for connecting this symbol to a hardware port. For this, double-click the symbol. The TYPE attribute is used to define the variable type (BIT, UBYTE, SBYTE, UINT, SINT or LONG). The COMMENT attribute is used in the report file.
	**inputlatch**	**Inputlatch Symbol.** The input value (name) is loaded when the enable input (En) is TRUE. You are prompted for the symbolic name and the type of input. The TYPE attribute is used to define the variable type (BIT, UBYTE, SBYTE, UINT, SINT or LONG). The COMMENT attribute is used in the report file. En type = boolean In type = Any type (BIT..LONG)
	**output**	**Output Symbol.** The NAME attribute is used for connecting this symbol to a hardware port. For this, double-click the symbol. The TYPE attribute is used to define the variable type (BIT, UBYTE, SBYTE, UINT, SINT or LONG). The COMMENT attribute is used in the report file.

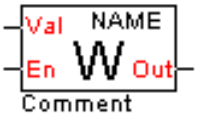
Symbol	Symbol Name	Description
	**outputlatch**	**Outputlatch Symbol.** The output value (Name) is transferred to the hardware port when the enable input (En) is TRUE. You are prompted for the symbolic name and the type of the physical output. The TYPE attribute is used to define the variable type (BIT, UBYTE, SBYTE, UINT, SINT or LONG). The COMMENT attribute is used in the report file. En type = boolean Out type = Any type (BIT..LONG)

8.2 Sequential Symbols

Sequential symbols allow you to sequence inputs and outputs. They are a means of sequencing signals in the main loop of the application, without introducing an interrupt.

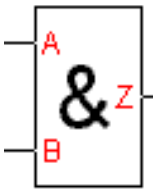
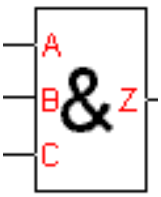
Symbol	Symbol Name	Description
	microdelf	**Micro Delay Fixed Symbol**. Delays the main loop for a fixed time. This symbol is meant for short and time-critical operations, and **does not** use the base clock timer tick, thereby leaving Timer 1 (ST6) or Timer A (ST7) free for other uses. En type = Boolean. When En is true (= 1) the fixed delay will become active. Out type = Boolean. The output is a copy of the En input. This output is used for linking the sequential symbols. ValueTime = The time, entered in microseconds, of the fixed delay/wait. **Note:** When the fixed delay is started, all normal execution process are halted. Only the interrupts are still active. The watchdog, if applicable, is not reloaded.

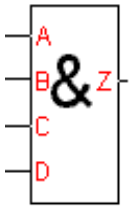
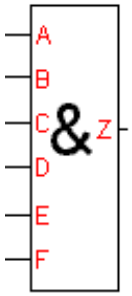
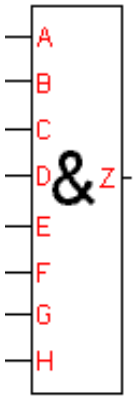
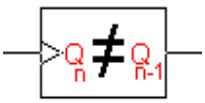
Symbol	Symbol Name	Description
	microdelv	**Micro Delay Variable Symbol**. Delays the complete process for a user defined time. This symbol is meant for short and time-critical operations. En type = Boolean. When true the fixed delay will become active. Time type = Word. Defines the time to wait. The Time input, together with the TimeBase, will determine the time to wait. Out type = Boolean. A copy of the En input. This output is used for linking the sequential symbols. TimeBase = The timebase, entered in microseconds. This time base is used a the timebase for the Time input. **Note:** When the variable delay is started, all normal execution process are halted. Only the interrupts are still active. The watchdog, if applicable, is not reloaded.
	**inputsequence**	**Sequential Input Symbol**. Converts, if enabled, a physical input to a binary value. This symbol is meant for time critical operations. Name = Physical input. The user is prompted for the symbolic name (In) and the input type. The connection is made to a physical micro controller pin or register and is invisible. En type = Boolean. When true the physical input is read. Out type = Boolean. A copy of the En input. This output is used for linking the sequential symbols. **Note:** When the input sequence is enabled, the physical input is read immediately. .
	outputsequence	**Sequential Output Symbol**. Converts, if enabled, a binary value to a physical output. This symbol is meant for time critical operations. Name = Physical output (type = All). The user is prompted for the symbolic name (OI). The connection is made to a physical micro controller pin and is invisible. En type = Boolean. When true the physical output is written. Out type = Boolean. A copy of the En input. This output is used for linking the sequential symbols. **Note:** When the output sequence is enabled, the physical output is written immediately.

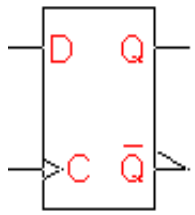
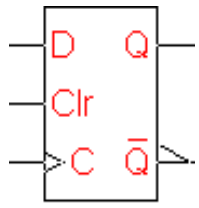
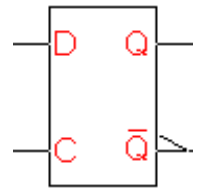
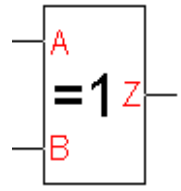
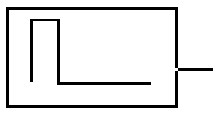
Symbol	Symbol Name	Description
	**waitsequence**	**Sequential Wait Symbol**. Reads a physical input and waits until it equals a pre-defined value. This symbol is meant for short and time critical operations. Name = Physical input (type = All). The user is prompted for the symbolic name (In). The input type equals the Val input type. The connection is made to a physical micro controller pin or register and is invisible. Val = The comparison value. En type = Boolean. When true the physical input is continuously read until the physical value equals the Val input. Out type = Boolean. A copy of the En input. This output is used for linking sequential symbols. **Note:** When the wait sequence is started, all normal execution process are halted. The physical input is read continuously until it equals the Val input. Only the interrupts are still active. The watchdog is re-loaded.

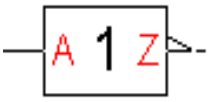
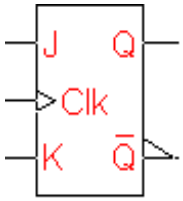
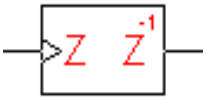
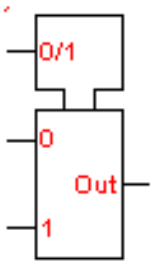
8.3 Logic Symbols

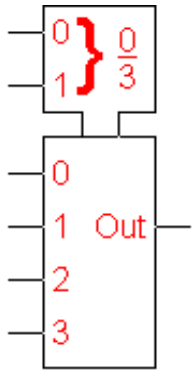
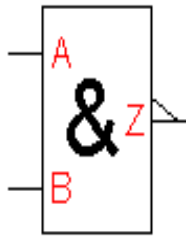
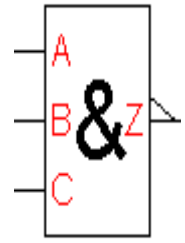
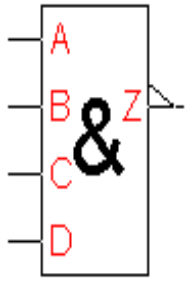
Logic symbols perform logical functions essential to most programs. With these functions, you can control a program's path, and depending on whether the logical condition is satisfied or not, divert the application into different subschemes or subdiagrams.

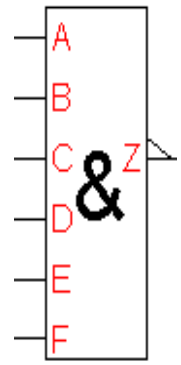
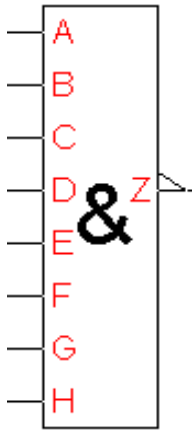
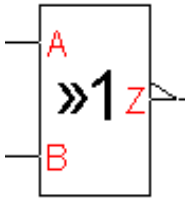
Symbol	Symbol Name	Description
	**and2**	**Two-value binary AND function symbol** with type inheritance. Z = A AND B A type = Any type (BIT..LONG) B type = Any type (BIT..LONG) Z type = The largest A or B type **Note**: ANDing a BIT-type value and a WORD-type value is not allowed.
	**and3**	**Three-value binary AND function symbol**, only applicable to BIT-type values. Z=A AND B AND C

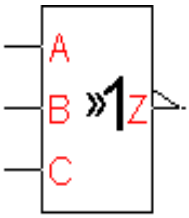
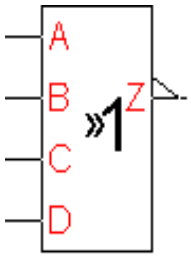
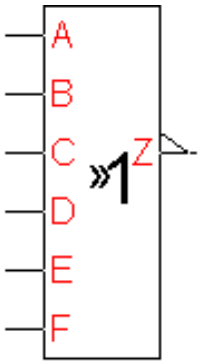
Symbol	Symbol Name	Description
	**and4**	**Four-value binary AND function symbol**, only applicable to BIT-type values. Z=A AND B AND C AND D
	**and6**	**Six-value binary AND function symbol**, only applicable to BIT-type values. Z=A AND B AND C AND D AND E AND F
	**and8**	**Eight-value binary AND function symbol**, only applicable to BIT-type values. Z=A AND B AND C AND D AND E AND F AND G AND H
	**change**	**Change Detection Symbol.** An output pulse is generated when the input value changes. IN type = Any type (BIT..LONG) OUT type = BIT

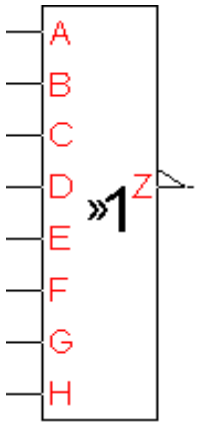
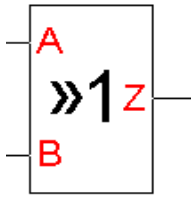
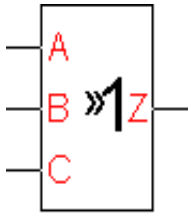
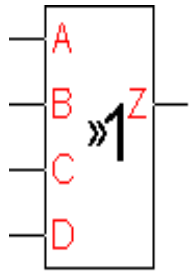
Symbol	Symbol Name	Description
	**dff** **pdff**	**D Element Symbol.** D flip-flop symbol with multiple-type data input. D input is clocked in on a rising edge on the C input. D input type = Any type (BIT..LONG) C input type = BIT Q, $\overline{Q}$ output types = same as D When in the persistent (non volatile) library this symbol (named **pdff**) shows an "**M**" in its design.
	**dff-clr**	**D Element with Clear Symbol.** D flip-flop symbol with multiple-type data and clear input. D input is clocked in on a rising edge on the C input. The D flip-flop is cleared when Clr is high. D input type = Any type (BIT..LONG) C input type = BIT Q, $\overline{Q}$ output types = same as D
	**dlatch**	**D Latch Symbol** with multiple-type data input. D input is clocked in when C input is high. D input type = Any type (BIT..LONG) C input type = BIT Q, $\overline{Q}$ output types = same as D
	**xor**	Two-value binary **Exclusive OR function symbol** with type inheritance. Z = A XOR B A type = Any type (BIT..LONG) B type = Any type (BIT..LONG) Z type = The largest A or B type **Note**: XORing a BIT-type value and a WORD-type value is not allowed.
	**init**	**Initial Loop Symbol.** Output TRUE only during the first pulse after reset. Out type = BIT

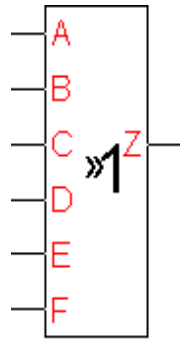
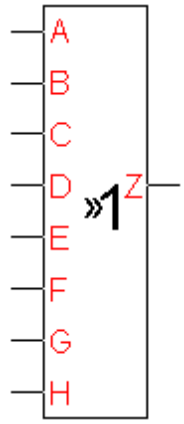
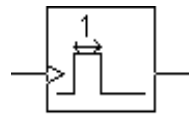
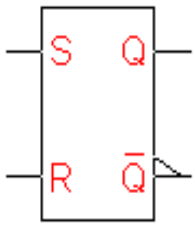
Symbol	Symbol Name	Description
	**inv**	Multi-type binary **Inverter Symbol**. Z= NOT A A type = Any type (BIT..LONG) Z type = A type
	**jkff**	**Jk Flip-flop Symbol**. Values are: Clk J K Q Q̄ 0 x x Q-1 Q̄-1 ! 0 0 Q-1 Q̄-1 ! 1 0 1 0 ! 0 1 0 1 ! 1 1 toggle Where: ! = rising edge x = don't care Q-1 = previous value
	**loopdel**	**One-loop Delay Symbol.** The output is the previous value of the current input. In type = Any type (BIT..LONG) Out type = In type
	**mux1**	**One-selection Input Multiplexer Symbol.** If the 0/1 selection input value is 0, Out takes the input 0 value. Otherwise Out takes input 1 value. 0/1 type = BIT 0, 1 type = Any type (BIT..LONG) Out type = The largest 0 or 1 type

Symbol	Symbol Name	Description
	**mux2**	**Two-selection Input Multiplexer Symbol.** Values taken by Out are shown in the following table: Sel bit 0 Sel bit 1 Out 0 0 Input 0 value 1 0 Input 1 value 0 1 Input 2 value 1 1 Input 3 value Sel 0/1 type = BIT 0, 1, 2, 3 type = Any type (BIT..LONG) Out type = The largest 0, 1, 2, or 3 type
	**nand2**	**Two-value binary NOT AND function symbol** with type inheritance. Z = NOT(A AND B) A type = Any type (BIT..LONG) B type = Any type (BIT..LONG) Z type = The largest A or B type **Note**: NANDing a BIT-type value and a WORD-type value is not allowed.
	**nand3**	**Three-value binary NOT AND function symbol**, only applicable to BIT-type values. Z= NOT (A AND B AND C)
	**nand4**	**Four-value binary NOT AND function symbol**, only applicable to BIT-type values. Z= NOT(A AND B AND C AND D)

Symbol	Symbol Name	Description
	**nand6**	**Six-value binary NOT AND function symbol**, only applicable to BIT-type values. Z= NOT(A AND B AND C AND D AND E AND F)
	**nand8**	**Eight-value binary NOT AND function symbol**, only applicable to BIT-type values. Z= NOT(A AND B AND C AND D AND E AND F AND G AND H)
	**nor2**	**Two-value binary NOT OR function symbol** with type inheritance. Z = NOT(A OR B) A type = Any type (BIT..LONG) B type = Any type (BIT..LONG) Z type = The largest A or B type **Note**: NORing a BIT-type value and a WORD-type value is not allowed.

Symbol	Symbol Name	Description
	**nor3**	**Three-value binary NOT OR function symbol**, only applicable to BIT-type values. Z= NOT (A OR B OR C)
	**nor4**	**Four-value binary NOT OR function symbol**, only applicable to BIT-type values. Z= NOT(A OR B OR C OR D)
	**nor6**	**Six-value binary NOT OR function symbol**, only applicable to BIT-type values. Z= NOT(A OR B OR C OR D OR E OR F)

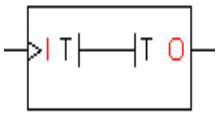
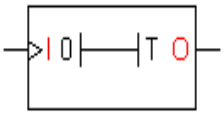
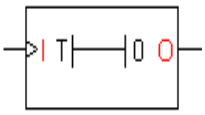
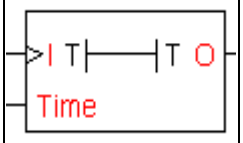
Symbol	Symbol Name	Description
	**nor8**	**Eight-value binary NOT OR function symbol**, only applicable to BIT-type values. Z= NOT(A OR B OR C OR D OR E OR F OR G OR H)
	**or2**	**Two-value binary OR function symbol** with type inheritance. Z = A OR B A type = Any type (BIT..LONG) B type = Any type (BIT..LONG) Z type = The largest A or B type **Note**: ORing a BIT-type value and a WORD-type value is not allowed.
	**or3**	**Three-value binary OR function symbol**, only applicable to BIT-type values. Z=A OR B OR C
	**or4**	**Four-value binary OR function symbol**, only applicable to BIT-type values. Z=A OR B OR C OR D

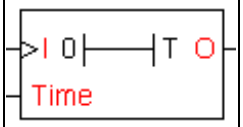
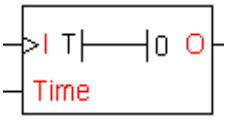
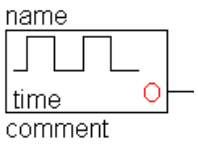
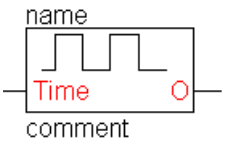
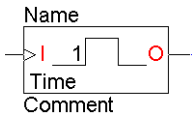
Symbol	Symbol Name	Description
	**or6**	**Six-value binary OR function symbol**, only applicable to BIT-type values. Z=A OR B OR C OR D OR E OR F
	**or8**	**Eight-value binary OR function symbol**, only applicable to BIT-type values. Z=A OR B OR C OR D OR E OR F OR G OR H
	**edge**	**Rising Edge Detector Symbol.** A rising edge on input causes an output pulse to be generated. In type = BIT Out type = BIT
	**srff** **psrff**	**Set/reset Flip-flop Symbol.** A high level on S causes output Q to be set to 1 ($\overline{Q}$ = 0) A high level on R causes output Q to be set to 0 ($\overline{Q}$ = 1) All types = BIT When in the persistent (non volatile) library this symbol (named **psrff**) shows an "**M**" in its design

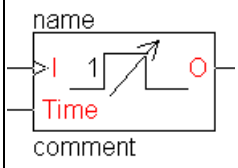
8.4 Time Related Symbols

Time related symbols perform functions where the concept of time is required. This range from requiring a delay period, or a timed operation, to providing an oscillating signal. These symbols can be used in the main loop and will run off of the default microcontroller timer. They do not interrupt the main loop.

Time-related symbols use the timer tick to evaluate whether their conditions in exactly the same way that periodic events do—refer to page 87 for a description of how periodic events evaluate time.

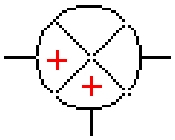
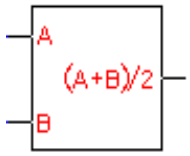
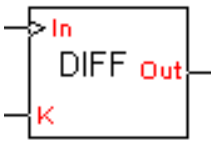
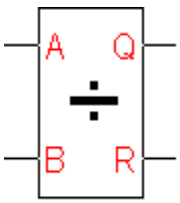
Symbol	Symbol Name	Description
	**delf**	**Delay Fixed Symbol.** An ON or OFF input bit is delayed a specified fixed time. The TIME attribute format is: `dd:hh:mm:ss.xxx` where: `dd` is the number of days, `hh` is the number of hours, `mm` is the number of minutes, `ss` is the number of seconds and `xxx` is a fraction of a second. I and O types = BIT
	**delfoff**	**Delay Fixed Off Symbol.** An OFF input bit is delayed a specified fixed time. The TIME value format is: `dd:hh:mm:ss.xxx` where: `dd` is the number of days, `hh` is the number of hours, `mm` is the number of minutes, `ss` is the number of seconds and `xxx` is a fraction of a second. I and O types = BIT
	**delfon**	**Delay Fixed On Symbol.** An ON input bit is delayed a specified fixed time. The TIME value format is: `dd:hh:mm:ss.xxx` where: `dd` is the number of days, `hh` is the number of hours, `mm` is the number of minutes, `ss` is the number of seconds and `xxx` is a fraction of a second. I and O types = BIT
	**delv**	**Delay Variable Symbol.** An ON or OFF input bit is delayed a variable time. The Time input defines the number of clock ticks for the delay (default= 10 ms). Time type = WORD I and O types = BIT

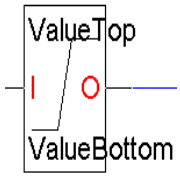
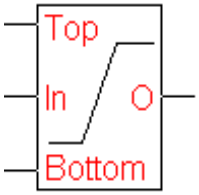
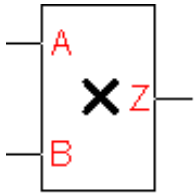
Symbol	Symbol Name	Description
	**delvoff**	**Delay Variable Off Symbol.** An OFF input bit is delayed a variable time. The Time input defines the number of clock ticks for the delay (default= 10 ms). Time type = WORD I and O types = BIT
	**delvon**	**Delay Variable On Symbol**. An ON input bit is delayed a variable time. The Time input defines the number of clock ticks for the delay (default= 10 ms). Time type = WORD I and O types = BIT
	**oscf**	**Fixed Time Oscillator Symbol.** The TIME attribute format is: `dd:hh:mm:ss.xxx` where: `dd` is the number of days, `hh` is the number of hours, `mm` is the number of minutes, `ss` is the number of seconds and `xxx` is a fraction of a second. Note that the frequency in Hz is = 1/(2*TIME) O type = BIT
	**oscv**	**Variable Time Oscillator Symbol.** The Time input defines the number of clock ticks (default= 10 ms). Note that the frequency in Hz is = 1/(2*TIME) TIME type = WORD O type = BIT
	**timf**	**Fixed Timer Symbol.** A rising edge on the input pin causes a pulse to be generated. The TIME attribute format is: `dd:hh:mm:ss.xxx` where: `dd` is the number of days, `hh` is the number of hours, `mm` is the number of minutes, `ss` is the number of seconds and `xxx` is a fraction of a second. I, O type = BIT

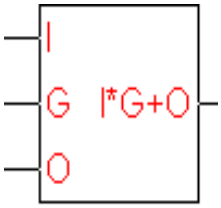
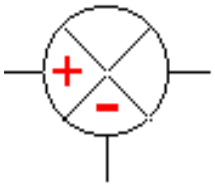
Symbol	Symbol Name	Description
	**timv**	**Variable Timer Symbol.** A rising edge on the input pin causes a pulse to be generated. The Time input defines the number of clock ticks for the pulse (default= 10 ms). Time type = WORD I, O type = BIT

8.5 Mathematical Symbols

These symbols perform mathematical functions such as adding, subtracting, multiplying, dividing, integrating and differentiating, so that you may treat and transform your application's signals.

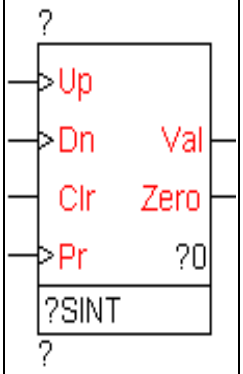
Symbol	Symbol Name	Description
	**add2**	**Two-value Add Symbol** with type inheritance. OUT =IN1+IN2 IN1 type = WORD IN 2 type = WORD OUT type = The largest IN1 or IN2 type
	**average**	**Average Symbol**. Takes average of IN1 and IN2. OUT = (IN1 + IN2)/2 IN1 type = WORD IN 2 type = WORD OUT type = The largest IN1 or IN2 type
	**differential**	**Differential Symbol.** Takes derivative of input (In) with respect to time. K is a constant input value. Out(t)= 2/3*(In(t)-In(t-dt))/dt+1/3*Out(t-dt) K=2/(3*dt) Type = depends on input type
	**div**	**Divider Symbol.** Q= A/B R = A%B A, B type = WORD Q, R type = The largest A or B type

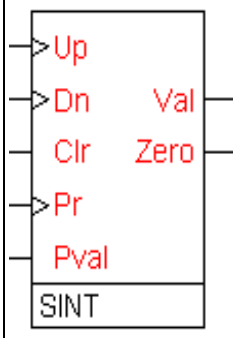
Symbol	Symbol Name	Description
	integral	**Integral Function Symbol.** Out(t) = Out(t-dt) + In(t)*dt K= 1/dt Type = depends on input type
	**limf**	**Fixed Limiter Symbol.** Output value is not larger than the top value, and not smaller than the bottom value. I type = WORD O type = I type
	**limv**	**Variable Limiter Symbol.** Output value is not larger than the top value, and not smaller than the bottom value. TOP type = WORD BOTTOM type = WORD In type = WORD O type = In type
	**mul**	**Multiplier Symbol.** Z = A*B A, B type = WORD Z type = The largest A or B type
	scalert	**Fixed Scaler Symbol.** G = Gain. O = Offset. Output = Input*Gain + Offset I type = WORD Output value type = I type

Symbol	Symbol Name	Description
	**scalerv**	**Variable Scaler Symbol.** G = Gain. O = Offset. Output = Input*Gain + Offset I type = WORD Output value type = I type
	**sub2**	**Two-value Subtractor Symbol** with type inheritance. OUT = IN1 - IN2 IN1 type = WORD IN 2 type = WORD OUT type = The largest IN1 or IN2 type

8.6 Counter Symbols

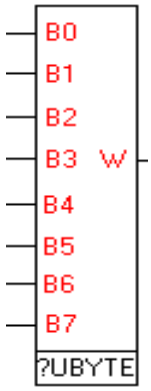
Counter symbols let you count the iterations of either a fixed or variable signal.

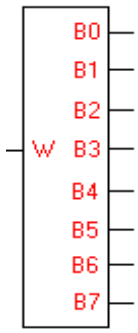
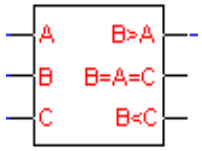
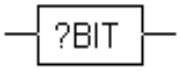
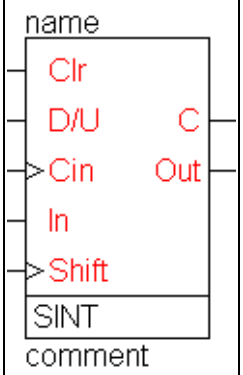
Symbol	Symbol Name	Description
	**countf** **pcountf**	**Counter symbol with fixed initial value.** A rising edge on the Up input pin causes the counter to be incremented by one. A rising edge on the Dn input pin causes the counter to be decremented by one. A high level on the Clr input pin causes the counter to be set to 0. A rising edge on the Pr input pin causes the counter to be loaded with the specified value (via the value dialog box). When the counter output value Val reaches 0 the Zero output value = 1. Up, Dn, Clr, Pr and Zero types = BIT Val type = WORD When in the persistent (non volatile) library this symbol (named **pcountf**) shows an "**M**" in its design

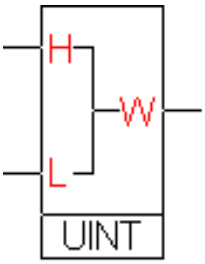
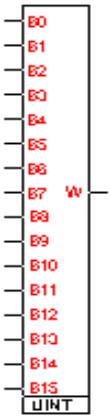
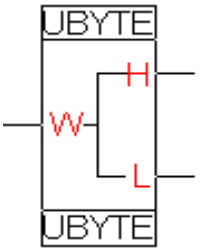
Symbol	Symbol Name	Description
	**countv** **pcountv**	**Counter symbol with variable initial value.** A rising edge on the Up input pin causes the counter to be incremented by one. A rising edge on the Dn input pin causes the counter to be decremented by one. A high level on the Clr input pin causes the counter to be set to 0. A rising edge on the Pr input pin causes the counter to be loaded with the Pval input value. When the counter output value Val reaches 0 the Zero output value = 1. Up, Dn, Clr, Pr and Zero types = BIT Val and Pval types = WORD When in the persistent (non volatile) library this symbol (named **pcountv**) shows an "**M**" in its design

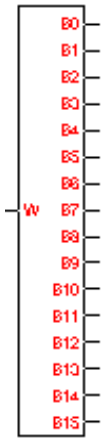
8.7 Conversion Symbols

Conversions symbols allow you to convert between different data types within the application.

Symbol	Symbol Name	Description
	**bpack**	**8-bit to 1-byte Packer Symbol.** Enables you to construct a byte out of eight bits. B0 to B7 type = BIT W type = WORD

Symbol	Symbol Name	Description
	**bunpack**	**1-byte to 8-bit Unpacker Symbol.** Enables you to unpack a byte into eight (or less) bits. W type = WORD B0 to B7 type = BIT
	**comp**	**Multi-purpose Comparator Symbol.** if B>A, the B>A output takes bit value =1 if B=A=C, the B=A=C output takes bit value =1 if B<C, the B<C output takes bit value =1 A, B, C types = WORD output types = BIT
	**convert**	**Type conversion symbol** that enables you convert a BIT type to a WORD type or a WORD type to a BIT type. IN type = BIT..LONG OUT type = BIT..LONG
	**shift** **pshift**	**Shift register symbol.** If Clr = 1, the shift register is cleared If D/U = 0, the shift direction is down If D/U = 1, the shift direction is up A rising edge on Cin causes the In input value to be loaded into the shift register. A rising edge on Shift triggers the shift operation in the shift register. The C output is the last bit shifted out (carry). Clr, D/U, Cin, Shift, C type = BIT In, Out type = Any type (BIT..LONG) When in the persistent (non volatile) library this symbol (named **pshift**) shows an "**M**" in its design

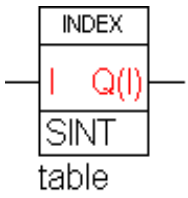
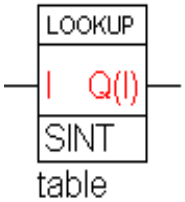
Symbol	Symbol Name	Description
	**wmerge**	**Word Merge Symbol.** Symbol used to build a word out of a high byte and a low byte. All types = WORD
	**wpack**	**Word Pack Symbol.** Symbol used to pack sixteen bits into one word.
	**wsplit**	**Word Split Symbol.** Symbol used to split an input word into a high byte and a low byte. All types = WORD

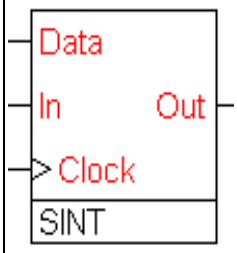
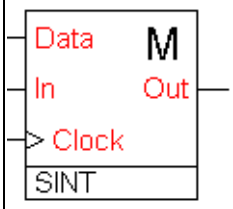
Symbol	Symbol Name	Description
	**wunpack**	**Word Unpack Symbol.** Symbol used to unpack a word into sixteen bits.

8.8 Table Symbols

Table symbols allow input values to be converted to output values according to a table of correspondence which you either input directly, or import from an ASCII file.

Note: Take care to choose the data input type (I type) carefully as some types use more of the microcontroller's available RAM than others.

Symbol	Symbol Name	Description
	**indextable**	**Index table Symbol.** The input value is used as an index in the table. The table is stored as an ASCII file defined by the TABLE attribute. Each table begins with a default value that is used when the input value is out of range. To create or change a table, double-click the symbol. I type = WORD Q(I) type = Any type (BIT..LONG) See also: **pitable** (**persistent index table**)
	**lookuptable**	**Lookup Table Symbol.** The input value is the search argument to find the output value. The table is stored as an ASCII file defined by the TABLE attribute. Each table begins with a default value that is used when the input value is out of range. To create or change a table, double-click the symbol. I type = WORD Q(I) type = Any type (BIT..LONG)

Symbol	Symbol Name	Description
	**ramtable** (volatile RAM)	**Ramtable Symbol.** Converts the input to a non-linear output value, found within the table. The input is used as an index to address the constant value from the table. You can change the table by clocking in the **Data** input at the position pointed to by the **In** input. Data, In type = WORD BlockSizeInUnits = size of the table.
	**pitable** (persistent library only)	**Index table symbol** for a table stored in persistent (non volatile) memory. The input value is used as an index in the table. You can change the table by clocking in the **Data** input at the position pointed to by the **In** input. Data, In type = WORD Q(I) type = Any type (BIT..LONG) See also: **indextable**

8.9 Power Management

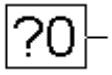
Power management symbols allow you to control the power consumption of the microcontroller by changing between normal, slow, stopped and wait modes.

Symbol	Symbol Name	Description
	slow	**Slow mode symbol.** Sets the microcontroller clock frequency lower. Refer to your target microcontroller's datasheet for information on controlling its clock frequency settings.
	stop	**Stop mode symbol.** Sets the microcontroller to standby. Requires a reset or external interrupt to wake up the microcontroller.

Symbol	Symbol Name	Description
	wait	**Wait mode symbol.** Sets the microcontroller to sleep mode. The microcontroller can be woken up by any interrupt.

8.10 Constant Symbols

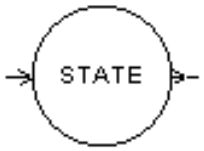
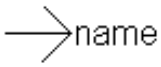
Constant symbols are useful when you need to compare a signal value to a fixed value.

Symbol	Symbol Name	Description
	**constb**	**Constant bit symbol.** Values 0 and 1 are allowed. OUT type = BIT
	constw	**Constant word symbol.** The value can be specified in the range -2147483648 to 2147483647. Values can be in decimal, binary, hexidecimal or octal types. The value is a decimal value by default. OUT type = WORD (determined by the value specified)

8.11 State Machine Symbols

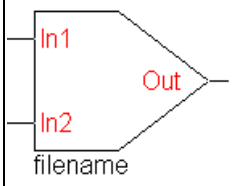
State machine symbols are used to create state machines for the application. State machine symbols define and control the progression of, the various states of your application.

Symbol	Symbol Name	Description
	stateinit **pstateinit**	**Initial state symbol** in a state machine. Every state machine must have one and only one **stateinit** symbol. This symbol can only be connected to **condition** or **state** symbols. When in the persistent (non volatile) library this symbol (named **pstateinit**) shows an "**M**" in its design .

Symbol	Symbol Name	Description
	**state**	**State symbol** used within a state machine. Can be used in conjunction with a **stateout** symbol within a scheme. See also **stateout**. The state symbol can only be connected to **condition**, **state** or **stateinit** symbols within a state machine.
	condition	**Condition function symbol** for state machines.This symbol can only be connected to **state** or **stateinit** symbols in state machines.The NAME attribute is used to make a connection with a **statein** symbol in schemes. See also **state**.
	**statein**	**State input symbol.** This symbol connects a BIT variable to a **condition** symbol within a state machine. The connection is performed by assigning the **condition** and **statein** symbols the same name. In type = BIT
	stateout	**State output symbol.** This symbol extracts a state from a state machine and converts it to a BIT variable. The connection is performed by assigning the **state** and **stateout** symbols the same name. See also **state**. Out type = BIT

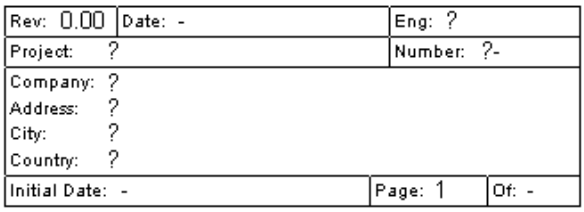
8.12 Hierarchical Sheet Symbols

Hierarchical sheet symbols are used to represents subschemes and their inputs and outputs.

Symbol	Symbol Name	Description
	portin	**Portin Symbol.** This symbol is used to connect the subscheme symbol pins from the parent scheme with the subscheme nets. The LABEL attribute is used to make the connection between the subscheme symbol pin and this symbol.
	portout	**Portout Symbol.** This symbol is used to connect the subscheme symbol pins from the parent scheme with the subscheme nets. The LABEL attribute is used to make the connection between the subscheme symbol pin and this symbol.
	**sssx_y**	**Subscheme symbol**, with x input pins and y output pins. The symbol column shows an example of a subscheme with 2 input pins and 1 output pin. The SCHEME attribute defines the filename of the subscheme. **portin** and **portout** symbols are used to make the connection between the actual subscheme and the pins of the subscheme symbol. Several types of subscheme symbols are available in the main library: **sss0_1**, **sss1_0**, **sss1_1**, **sss1_2**, **sss2_1**, **sss2_2**.

8.13 Title Symbols

The title symbol is used to enter information about the scheme you are working on, for archiving purposes.

Symbol	Symbol Name	Description
	**title**	**Title block** used for archiving purposes. There are numerous fields that you can enter values for to record information about the scheme, the circumstances of its creation and the project.

9 ANALYSING AND GENERATING YOUR APPLICATION

9.1 Overview

ST-Analyser is a built-in tool that analyses your application for errors and converts it into either ST6 or ST7 code, depending on the target microcontroller. If any errors are encountered during the analysis and code generation, online information is provided to help you locate the error and take corrective action.

When you execute the analysis and compile, ST-Analyser analyses the current project by creating the netlist and cross references, then analysing and generating the final code. During this process, all project schemes are checked for connections between symbols, I/O assignments and variable types, before generating the source code. Providing no fatal errors are encountered, ST-Realizer generates a non-compiled ST macro-assembler language (**.asm**) file from the scheme.

ST-Analyser then generates the binary ST code. This process runs the STMicroelectronics ST assembler. Depending on whether or not you included the ROS (see Section 2.3 on page 5 and Section 5.3.4 on page 63), a file with extension ***.hex** or ***.obj** respectively is generated for ST6, or with extension ***.s19** or ***.obj** for ST7. A ***.hex** (or ***.s19**) file can be directly loaded into an ST MCU while you must link a ***.obj** file with another program.

When the analysis process has been successfully completed, a report file is generated. This report file gives information about the designation of I/O pins, a list of the variables used by type and the memory space required by the application.

If any messages are generated during the analysis and compile, you can view them and trace them back to their origin.

ST-Realizer includes hardware-specific characteristics, which you must configure prior to analyzing and compiling your code. They cover general hardware, memory, and peripheral information. See Section 5.3 on page 62 for further details.

9.2 Changing the Compile Options

Before you analyse and compile schemes using ST-Realizer, you can set the following compilation options:

- Whether or not you want to generate source code.
- Whether or not you want to invoke the assembler.
- Whether or not you want to generate final hex code.
- Whether or not you want to include project data (this can be useful for version control).
- Whether or not you want to generate code that uses a constant sampling time (you must

choose this option if your scheme includes the mathematical symbols: integral and differential). If you choose this option, the duration of the main loop is constant.

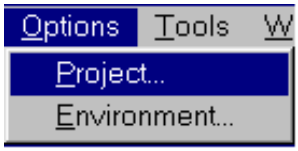
- The frequency of the base clock (tick) for the timer symbols. By default this is 0.01 seconds (10 milliseconds).

To generate, in a single run, executable code that can be embedded in an ST device, you must use the default settings.

Each ST device needs its own assembler and target compiler to create the final executable file. These elements are standard components of the ST-Realizer system.

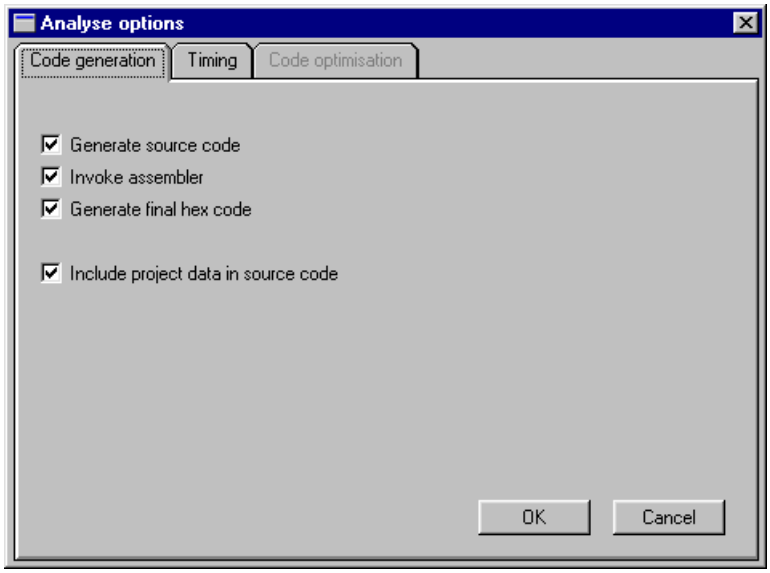
To view or set the compile options:

1 On the main menu, click *Options*, then *Project*...



The **Analyse options** dialog box opens, with tabs for you to select daughter dialog boxes.

2 The Code Generation dialog box is the first of the tabs.



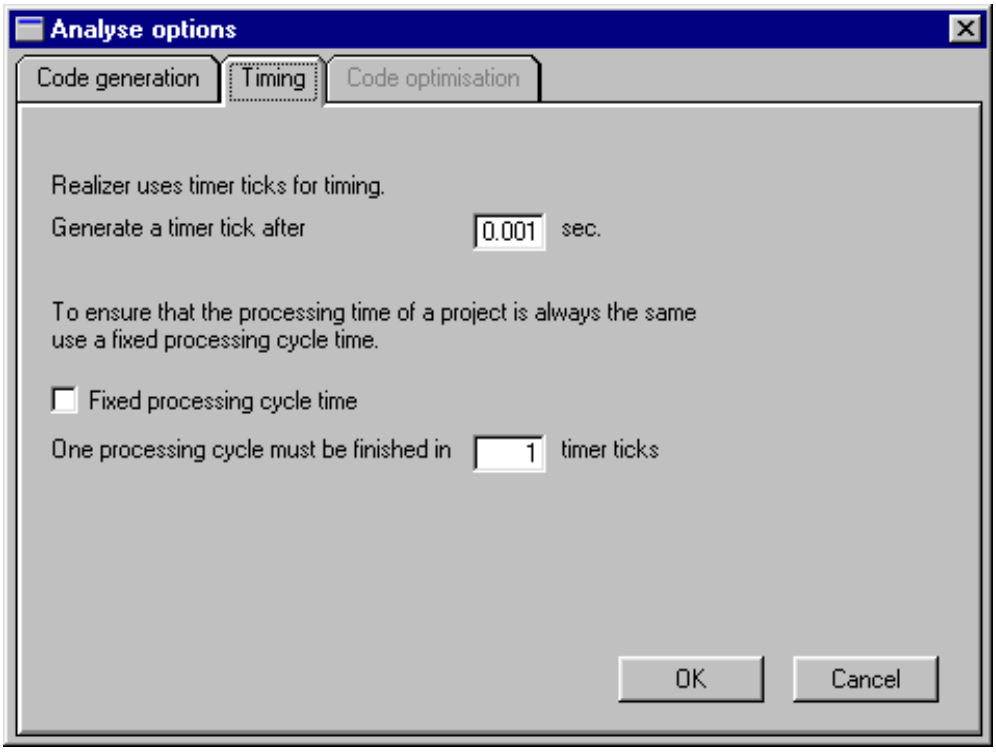
If you deselect the ***Generate source code*** option, the ***Invoke assembler*** and ***Generate final hex code*** options will also be deselected, as these are suboptions of the code generation process. They control, respectively, the generation of your application in ST6 or ST7 assembler code (the creation of the **`.asm`** file), and the creation a binary (hex) version of the code (a **`.hex`**, **`.obj`** or **`.s19`** file, depending on the microcontroller as described on page 129).

Tip: *Note that you must generate the assembler version before you can produce the binary (hex) code. The first three options must be selected in order to produce the binary (hex) code you need to load your application into your microcontroller using a programming tool.*

The final option in the **Code generation** dialog box is to include project data in the source code. If this option is enabled, the Analyse report will record how many times Analyse has been performed, which is useful in keeping track of different project versions.

Once you have selected the options you wish, click OK.

3 The second tab opens the Timing dialog box.



Every ST6 or ST7 microcontroller has a timer, called Timer 1 (ST6) or Timer A (ST7), which (if there are either time-related symbols or events in the application) is used as the base clock to measure out units of time called “timer ticks”. Timer ticks are the smallest increment of time that a timer can count. In this dialog box, you may choose the value for the “timer tick” or base clock unit used in the application.

The minimum value that you can enter for the timer tick depends on the target hardware device. The table below summarizes the minimum and maximum values for the timer tick

for each family of microcontroller.

	ST6 (8 MHz crystal)	ST7 (16 MHz crystal)
Minimum Timer Tick	100 µs	75 µs
Maximum Timer Tick	48 000 µs	60 000 µs

Timer ticks may also be used to count the number of main loop cycles. The time it takes to perform a main loop cycle is called the ***Processing Cycle Time***. By default, the processing cycle time is variable, with the shortest (fastest) possible processing cycle time being 1 timer tick. However, in this dialog box, you can choose to ***fix*** the processing cycle at a specific number of timer ticks of your choosing, by selecting ***Fixed processing cycle time***, and then specifying the number of timer ticks that you wish to correspond to a single processing cycle.

Note: Timer ticks (and the code needed to generate them) are **ONLY** created if you use either time-related symbols, or events that require a timer, in your application. Otherwise, the processing cycle time will be variable.

If your application uses any symbols or events that require the measurement of time (the timer symbol, for example) the default timer (Timer 1 or Timer A) will be used to measure that time using timer ticks. In addition, all time-related symbols or events (such as a periodic event) that are based on the timer tick, can also be controlled using the default timer. However, if you require a further timer that will **NOT** be based on the timer tick (as is the case, for example, when you wish to generate a pulse width modulated (PWM) signal), you will have to enable one of the peripheral timers on the microcontroller (if it exists—make certain that the microcontroller you have chosen can provide all of the peripherals you require).

To give an example of how the timer tick is used, imagine that you decide that you set the value of the timer tick at 0.01 seconds. When you start running your application, the default timer (Timer 1 or Timer A) will count out the number of ticks (each tick being equal to 0.01 s). At the beginning of the second main loop cycle, the total number of timer ticks that have elapsed during the first main loop cycle is recorded (i.e. the number of elapsed timer ticks is written to a variable called “rtick”). This recorded value is used to evaluate all [1]

1. Some time-related events use the timer tick differently to evaluate elapsed time more precisely. For more information, refer to “Timed interrupts” on page 88.

second main loop cycle. Simultaneously, the timer resets and begins counting the number of ticks that elapse during the running of the second main loop cycle. At the beginning of the third main loop cycle, the number of timer ticks that have elapsed during the running of the second main loop cycle is recorded, and this recorded value is used during the execution of the third main loop cycle.

If you choose to fix the processing cycle time, the number of timer ticks elapsed during each main loop cycle remains constant.

Tip:

Please note that there is one golden rule when it comes to timers—the quantity of time that you are measuring (either using a timer symbol or a periodic event) must be substantially greater than one tick. Because these time-related functions are based on the timer tick, they cannot measure times less than the value of one tick. Furthermore, the closer the value of the measured time period is to the value of one tick, the more inaccurate the timer function will be (refer to "Periodic Events" on page 87 for details on why this is).

Once you have entered the Timer options you wish, click OK.

4 The last tab opens the *Optimisations* dialog box.

The options in this dialog box are not currently available in this version of ST-Realizer II. However they are implemented in Realizer Gold which can be obtained from Actum Solutions.

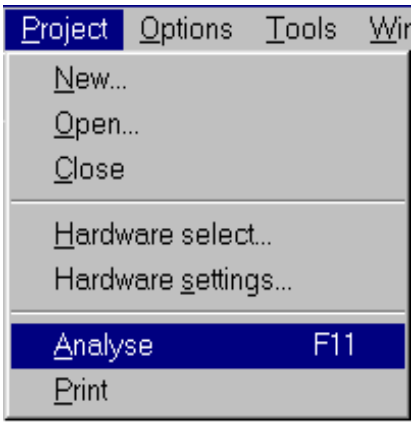
9.3 Executing the Analysis and Compile

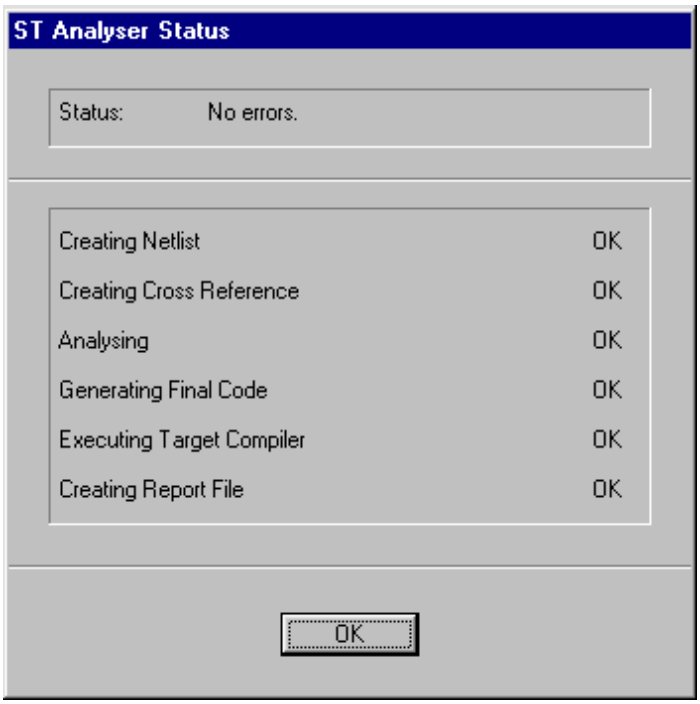
When you execute the analysis and compile, ST-Realizer performs the following tasks:

- Creates a netlist of all the schemes in the project.
- Creates the cross references between all schemes in the project.
- Analyses the logical functionality of the schemes for consistency.
- Generates the ST code.
- Transfers the ST code to the assembler for the chosen target hardware. This assembler generates the final executable code that is transferred into the target ST.
- Scans the map file and generates a report file.

You may execute the ST-Analyser from any of the project windows.

- In the **Project menu**, simply click ***Analyse***. The **ST Analyser Status** window will open and you will see the Analyse function checking your application step-by-step.





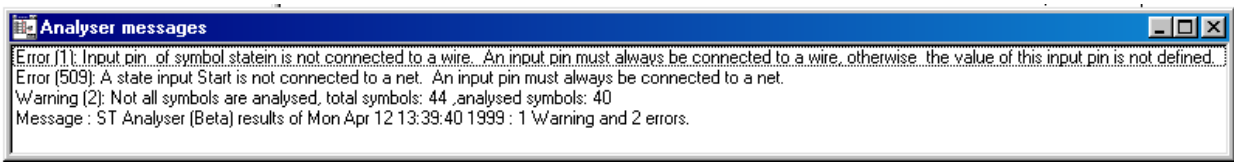
- When the analyse process has finished successfully (with no errors), click OK at the bottom of the **ST Analyser Status** window.
- A new window will appear on your screen, entitled **Analyser messages**. It will simply report the date and time and that “no errors” were found.

9.4 What to Do if there are Errors Found during Analyse

1 If errors are found, the **ST Analyser Status** window will show a “fail” message and give the number of errors found. Click OK.

2 A descriptive list of the errors will be shown in the **Analyser messages** window which then opens at the bottom of the scheme window. You can view each error by either double-clicking on its message in the window, or by clicking the previous message or next

message icons on the toolbar. The area of the scheme where the error occurs will appear in the scheme window.

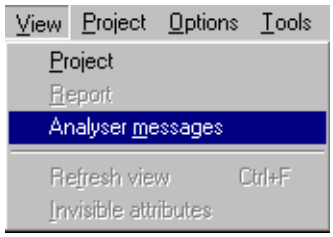

Analyser messages
Error (1): Input pin of symbol statein is not connected to a wire. An input pin must always be connected to a wire, otherwise the value of this input pin is not defined.
Error (509): A state input Start is not connected to a net. An input pin must always be connected to a net.
Warning (2): Not all symbols are analysed, total symbols: 44 ,analysed symbols: 40
Message : ST Analyser (Beta) results of Mon Apr 12 13:39:40 1999 : 1 Warning and 2 errors.

3 Correct the errors and re-compile the application by clicking on ***Analyse*** under the **Project** menu.

9.5 Viewing and Tracing Generated Messages

The information in the **Analyser messages** window is saved after you run ***ST-Analyser*** and can be displayed at any time:

1 Click **,**



or:

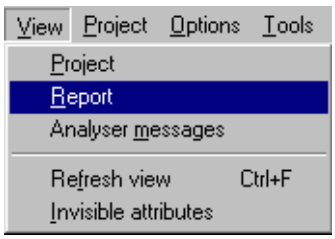
Select *Analyser messages* in the View menu:

The **Analyser messages** window for the last analysis opens.

2 Double-click any displayed message to view its origin.

9.5.1 Viewing the Analyse and Compile Report

Once you have analysed your scheme and compiled your program code, you can view the report generated by ST-Realizer during the analysis and compilation process. This report provides you with useful information such as the input and output connections you made, and gives an overview of how much memory is used by the application.



To see the report, on the **View** menu, click **Report**. An example of an application report is shown on the following pages.

```
ST72212G2 Realizing Unit (V4.00) (c) 1990-98 Actum Solutions
Report file of project C:\Program Files\ST-Realizer\Examples\Heating\heating.rpf
Scheme Version   : 1.00
Report timestamp : Wed Apr 14 13:14:40 1999
Analyze results  : No errors
-----------------------------------------------------------------


Schematic dependencies and events:
-----------------------------------------------------------------
C:\Program Files\ST-Realizer\Examples\Heating\heating.sch
Scheme: C:\Program Files\ST-Realizer\Examples\Heating\Filter.sch
Event:  Timed interrupt 0.01 Sec.

ST72212G2 (DIL28) connection overview:
-----------------------------------------------------------------
 Pin    Name   Alternative name Type I/O          Description
  1:    RESET                 (BIT   Input ),     Active low
  2:    OSCin                 (             ),    Oscillator In
  3:    OSCout                (             ),    Oscillator Out
  4:    PB.7   SS             (BIT   Input ),     Not connected
  5:    PB.6   SCK            (BIT   Input ),     Not connected
  6:    PB.5   MISO           (BIT   Input ),     Not connected
  7:    PB.4   MOSI           (BIT   Input ),     Not connected
  8:    PB.3   OCMP2_A        (BIT   Input ),     Not connected
  9:    PB.2   ICAP2_A        (BIT   Input ),     Not connected
 10:    PB.1   OCMP1_A        (BIT   Input ),     Not connected
 11:    PB.0   HeatingIsON    (BIT   Output),     Push-pull output
 12:    PC.5   Temperature    (UBYTE Input ),     8 bit analog input
 13:    PC.4   Setpoint       (UBYTE Input ),     8 bit analog input
 14:    PC.3   ICAP2_B        (BIT   Input ),     Used by the application
 15:    PC.2   CLKOUT         (BIT   Input ),     Not connected
 16:    PC.1   OCMP1_B        (BIT   Input ),     Used by the application
 17:    PC.0   ICAP1_B        (BIT   Input ),     Used by the application
 18:    PA.7                  (BIT   Input ),     Not connected
 19:    PA.6                  (BIT   Input ),     Not connected
 20:    PA.5                  (BIT   Input ),     Not connected
 21:    PA.4                  (BIT   Input ),     Not connected
 22:    PA.3                  (BIT   Input ),     Not connected
 23:    PA.2                  (BIT   Input ),     Not connected
 24:    PA.1                  (BIT   Input ),     Not connected
 25:    PA.0                  (BIT   Input ),     Not connected
 26:    TEST                  (             ),    Test mode pin
 27:    Vss                   (             ),    Ground
 28:    Vdd                   (            ),     Power Supply


Hardware connections:
-----------------------------------------------------------------------
Symbolic name H/W nameDescription          | Comment
HeatingIsON   PB.0       Push-pull output  | Pilot LED ON
Setpoint      PC.4 8 bit analog input      | Temperature in 1/10 degrees Celsius
Temperature   PC.5 8 bit analog input      | Temperature (in 1/10 degrees Celsius
```

```
Register connections:
-----------------------------------------------------------------------
Symbolic name H/W nameDescription                   | Comment
ARCP          TBOC1LRTimerB output compare 1 low re| PumpSpeedControl

List of all used peripherals:
-----------------------------------------------------------------------
TIMERB settings:
Prescaler value = 1/2
Output level 1 = Low
Output level 2 = High
Capt1 transition = Rising
Capt2 transition = Falling
Output Compare 1 = Enabled
Output Compare 2 = Disabled
Forced Output Compare 1 = Disabled
Forced Output Compare 2 = Disabled
Initialise compare register 1 = 000H
Initialise compare register 2 = 100H
PWM = Enabled
One Pulse Mode = Disabled

Variable overview:
-----------------------------------------------------------------------
Total used bits              : 13
Total used events            : 1
Total used unsigned bytes    : 7
Total used signed bytes      : 0
Total used unsigned integers : 18
Total used signed integers   : 3
Total used longs             : 0


Memory overview:
-----------------------------------------------------------------------
Total used RAM               : 63 byte (0080H,0081H->00BEH)
Total used ROM               : 892 byte (E000H->E37BH) of FFDFH
```

Note: Note that if you add up the RAM memory used in the variable overview and compare it with the value shown in the memory overview, there is a disparity. This is because the Total RAM includes memory for internal variables used by ST-Realizer.

9.6 Printing Reports

To print a report:

1 **Display the report.**

2 **Select File→Print in the main menu bar.**

3 **Continue the normal printing dialog and click OK when ready.**

10 SIMULATING YOUR APPLICATION

Once you have designed and analysed your application, you can use ST-Simulator to simulate its behavior, generate and view input signals, monitor signals that are generated by your application, and fine-tune it if necessary. The principal tasks involved in simulating an application are:

- Creating a simulation environment file, which defines the environment in which you'll simulate your application.
- Adding adjusters to your simulation environment file, which enable you to view and control the value of the input signals to your application.
- Adding probes to your simulation environment file, which enable you to view the signals that are generated by your application.
- Running the simulation.

You can also record the values generated by adjusters and read by probes while a simulation is being run. ST-Realizer records this information in Log files. This information can be useful for viewing the exact adjuster and probe values at any given time during the simulation.

This section explains how to perform these tasks.

10.1 Working with Simulation Environment Files

Each project you want to simulate must have its own simulation environment (**.sef**) file before it can be simulated. A simulation environment file contains copies of the project schemes with adjusters and probes added where required. You can save and reuse simulation environment files as you can do with any other type of file.

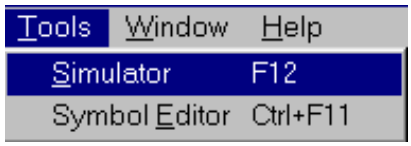
10.1.1 Creating a New .sef File

The first step in simulating your application is to create an associated simulation file. Simulation files are based on schemes, and are created by ST-Simulator.

To create a new simulation environment file:

1 **If you are in ST-Realizer, make sure that the project you wish to simulate is open.**

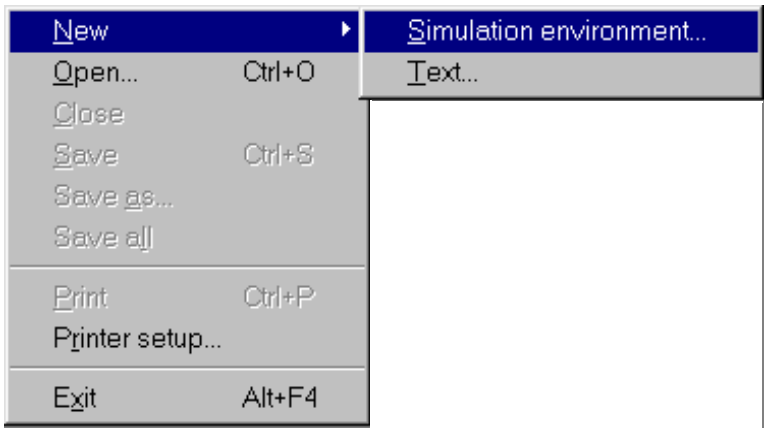
2 **Click Simulator on the ST-Realizer *Tools* menu.**



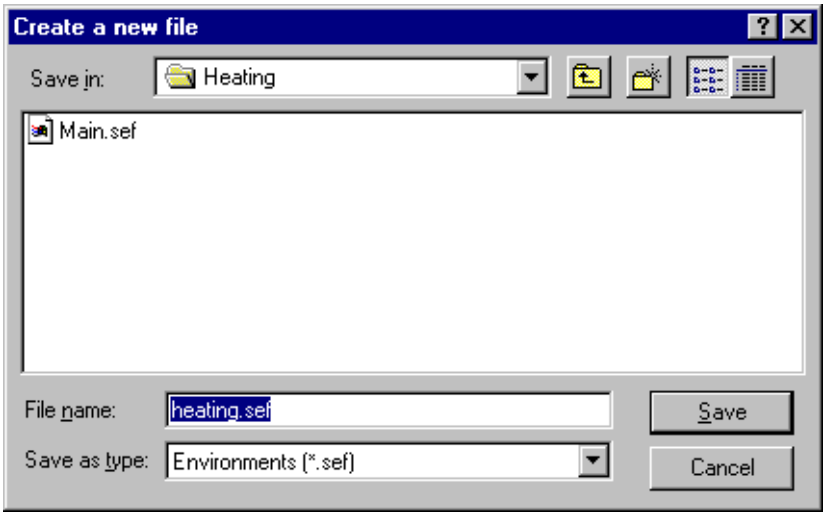
ST-Simulator will open in a separate window.

3 In ST-Simulator, select the *File* menu and click *New*.

In the cascading menu that appears, select ***Simulation environment***.



The ***Create a file*** dialog box opens, letting you specify the name of your new simulation environment (**`.sef`**) file.



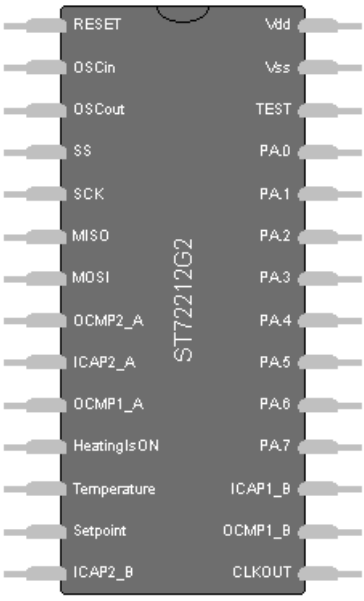
4 Type the name of the .sef file, (*heating.sef*, for example).

A pin level drawing of the target microcontroller appears (such as shown at right). Note that where the pins have been connected to application input/output functions, the pin names have been replaced by the function names.

If you double-click on the pin level drawing, you will open a copy of your root scheme diagram as another window. You can also open subschemes by double-clicking on the subscheme symbols.

You can use both the pin level drawing and the scheme diagrams to help you run the simulation—both views are useful in evaluating whether your application is running as you wish.

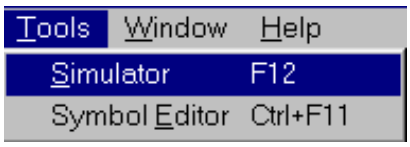
To see all views at once, under the *Window* menu, select *Tile*.



You've now created a new simulation environment file for your application. Don't forget to save your work (see Section 10.1.3 below).

10.1.2 Opening an Existing `.sef` File

1 If you are already running ST-Realizer, open the Simulator Window by clicking on Simulator on the ST-Realizer *Tools* menu.



Tip: *You can access ST-Simulator directly from Windows by selecting* ***Start→Programs→ST-Realizer II→Simulator.***

2 Once you are in ST-Simulator, select the *File* menu and click *Open*.

The Open simulation environment dialog box opens:

3 In the list, double-click the name of the `.sef` file (for example *main.sef*), then click *Open*.

All of the simulation views contained in the `.sef` file will open (for example, the pin-level drawing of the target microcontroller, and/or the project schemes). You are free to run or modify the simulation at will.

10.1.3 Saving an .SEF File

You should save the modifications you made on a **.sef** file at regular intervals and, of course, once you have finished all your work on the file. You may also make a duplicate of an existing **.sef** file, under a different filename, if you wish to have several different simulation versions.

To save an .sef file:

- On the **File** menu, click ***Save***.

To make a duplicate of an existing .sef file:

1 **On the File menu, click *Save as.***

The **Save file as** dialog box opens.

2 **Browse to the folder in which you want to create the duplicate file.**

3 **Type the new file name in the *File name* field, then click OK.**

Note: You may save your file as a .WMF (Windows Meta File)-type file. This will enable you to export the drawing to any word processing or publishing project file that accepts this format.

10.2 Setting, Adjusting and Viewing Input Values

When you simulate an application, you can set, adjust and view the values input to the application. This enables you to experiment with and fine-tune your application.

To be able to set, adjust and view input values and types, you connect adjusters to the appropriate pins and wires in the pin-level and scheme views of your application. ST-Realizer lets you set four types of adjusters:

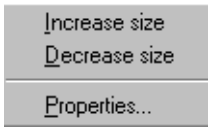
- **Numeric Adjusters**, which display and let you adjust variable values numerically, in binary, decimal, hexadecimal or octal format
- **Sine Wave Adjusters**, which generate and let you define analog sine wave signals. They let you adjust the wave depth and frequency.
- **Square Wave Adjusters**, which generate and let you define analog square wave signals. They let you adjust the wave depth, frequency, and duty cycle. The duty cycle is the percentage of the period in which the signal is at its top value.
- **Time Table Adjusters**, which let you adjust the value of a variable at specified time intervals.

The following paragraphs describe how to attach adjusters to pins and wires, and how to set the appropriate input values.

Note: 1. You can only attach one adjuster to each pin or wire.
2. Clicking any object in the schematic causes information on this object to be displayed in the status bar.

Changing Adjuster or Probe Attributes

If you right-click any type of adjuster or probe (probes are described in Section 10.3 on page 151), a popup menu will open that looks similar to that at right:



From this menu you may increase or decrease the size of the adjuster by selecting ***Increase size*** or ***Decrease size*****.**

10.2.4 Setting Fixed Input Values

To set a fixed analog or binary input value, you must attach a **Numeric Adjuster** to the appropriate input pin or wire. Numeric Adjusters let you set the value input by the pin or wire to which it is attached, as well as change the number base that the value is displayed in.

To attach a Numeric Adjuster to a pin or wire:

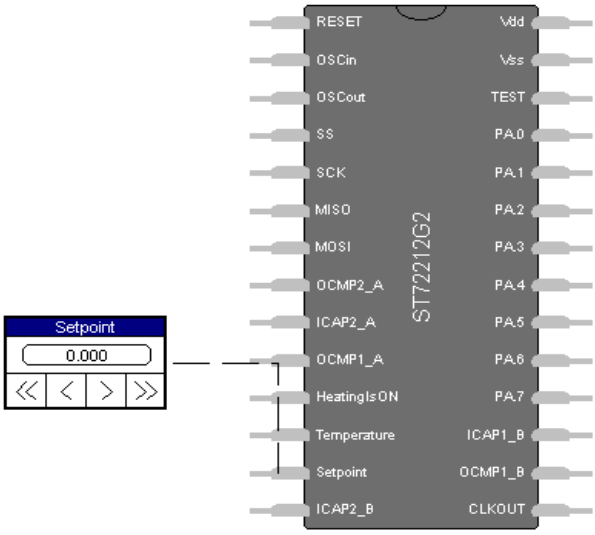
1 Select the pin or wire to which you want to attach the Numeric Adjuster by clicking on it.
The pin or wire you choose should be an input pin or wire. For example, select the pins on the pin-level drawing that you have attached to input symbols in your scheme (refer to “Connecting Input/Output Symbols to Microcontroller Pins, Ports and Peripherals” on page 80).

Note: A numeric adjuster has the same type as the pin or wire it is attached to.

2 Click **.**

3 Click where you want the Numeric Adjuster to appear.

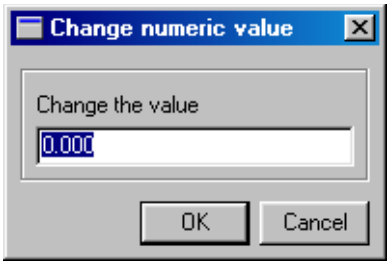
The adjuster is now placed in your scheme and connected to the pin or wire you selected. In the example at right, we see that the numeric adjuster has been connected to the pin called “Setpoint”, which is also the name of the input symbol. Note that the numeric adjuster shows the name of the pin whose input the adjuster controls.



4 Set the value input to the pin or wire.

If the pin or wire input is analog, you can change its value by either:

- Clicking the displayed value in the Numeric Adjuster and entering a new value in the **Change numeric value** dialog box, shown at right.
- Clicking the <<, <, > or >> buttons on the Numeric Adjuster:

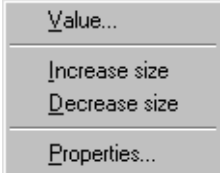


Numeric Adjuster Button	Action
<<	Decreases the input value by the fast step increment value (see below).
<	Decreases the input value by the step increment (see below).
>	Increases the input value by the step increment.
>>	Increases the input value by the fast step increment value.

- If the pin or wire input is binary, you can change its value by clicking the displayed value in the Numeric Adjuster icon. The value toggles between 0 and 1.

Tip:

*You may also open the **Change numeric value** dialog box to change the value of the numeric adjuster (as shown above), by right-clicking the adjuster and selecting **Value** from the popup menu at right.*



If you select the ***Properties*** option in the popup menu, a large dialog box will open, entitled **Change numeric adjuster**. From this dialog box, you can several numeric adjuster options, described in Table 5.

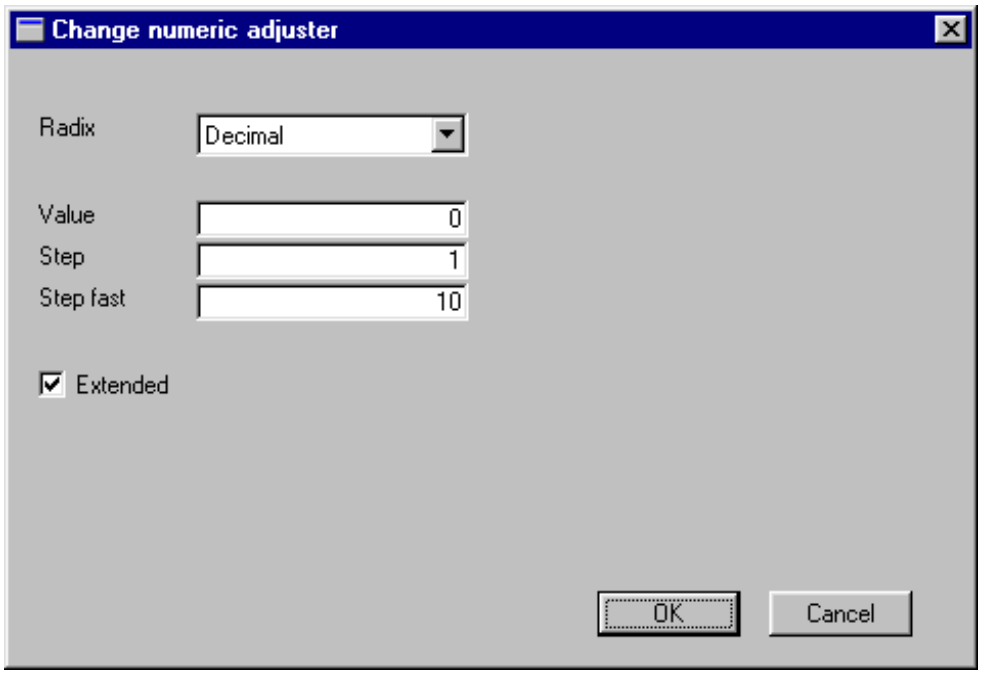


Table 5 Changing Numeric Adjuster Options

Option	Action
Change the number base in which the value is set and displayed.	Select the appropriate button in the **Radix** box.
Change the current value of the attached pin or wire.	Enter the new value in the **Value** field.
Change the increment value of the < and > buttons.	Enter the new value in the **Step** field.
Change the increment value of the << and >> buttons	Enter the new value in the **Step fast** field.
Hide or display the <, >, << and >> buttons.	Click the **Extended** check box. When this box is checked, the buttons are displayed.

10.2.5 Setting Variable Input Values

To set variable analog or binary input values, attach a **Time Table Adjuster** to the appropriate input pin or wire. The Time Table Adjuster lets you set an input value that changes at specified time periods. Once you have placed a Time Table Adjuster, you can either import data from an ASCII text file or enter the data directly into the Time Table Adjuster. The following diagram shows the format of Time Table Adjuster-compatible ASCII files:

ASCII File Contents:	**Table Values:**
5,100 75000,150 600000,75	

	time	output
0	00:00:00.0000	0
1	00:00:00.0005	100
2	00:00:07.5000	150
3	00:01:00.0000	75

You can store decimal, binary, hexadecimal or octal information in Time Table Adjusters.

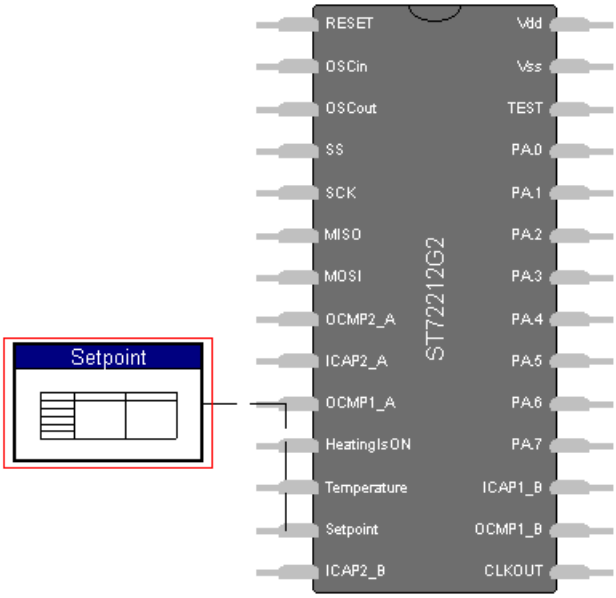
To attach a Time Table Adjuster to a pin or wire:

1 Select the pin or wire to which you want to connect the adjuster by clicking on it.

2 Click **.**

3 Click where you want the adjuster to appear.

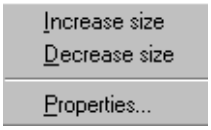
The adjuster is now placed in your scheme and connected to the pin or wire you selected. In the example at right, we see that the time table adjuster has been connected to the pin called “Setpoint”, which is also the name of the input symbol. Note that the time table adjuster shows the name of the pin whose input the adjuster controls.



4 To set the table of values to be input to the pin or wire:

- **Double-click the time table adjuster,**

or,



- **Right-click the time table adjuster, and select *Properties* in the popup menu.**

The Change time table adjuster dialog box opens, showing a table with one row, and two columns—the first for the time, and the second for the input value. From this dialog box, you can either import them from an ASCII file or enter the table values directly (see Table 6 for instructions).

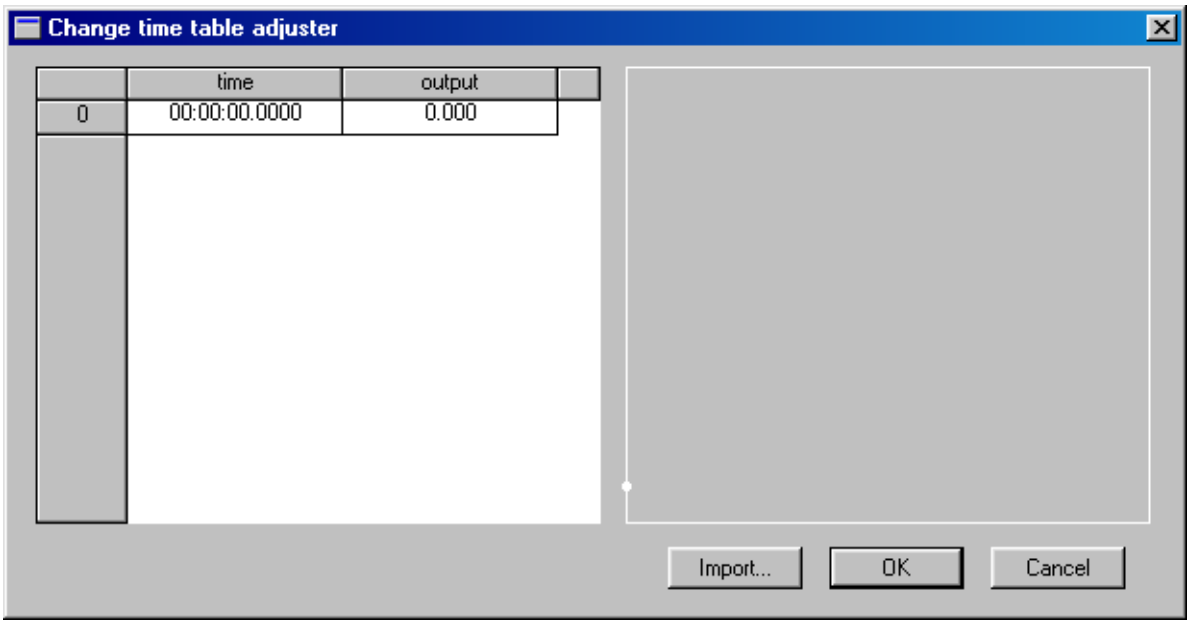


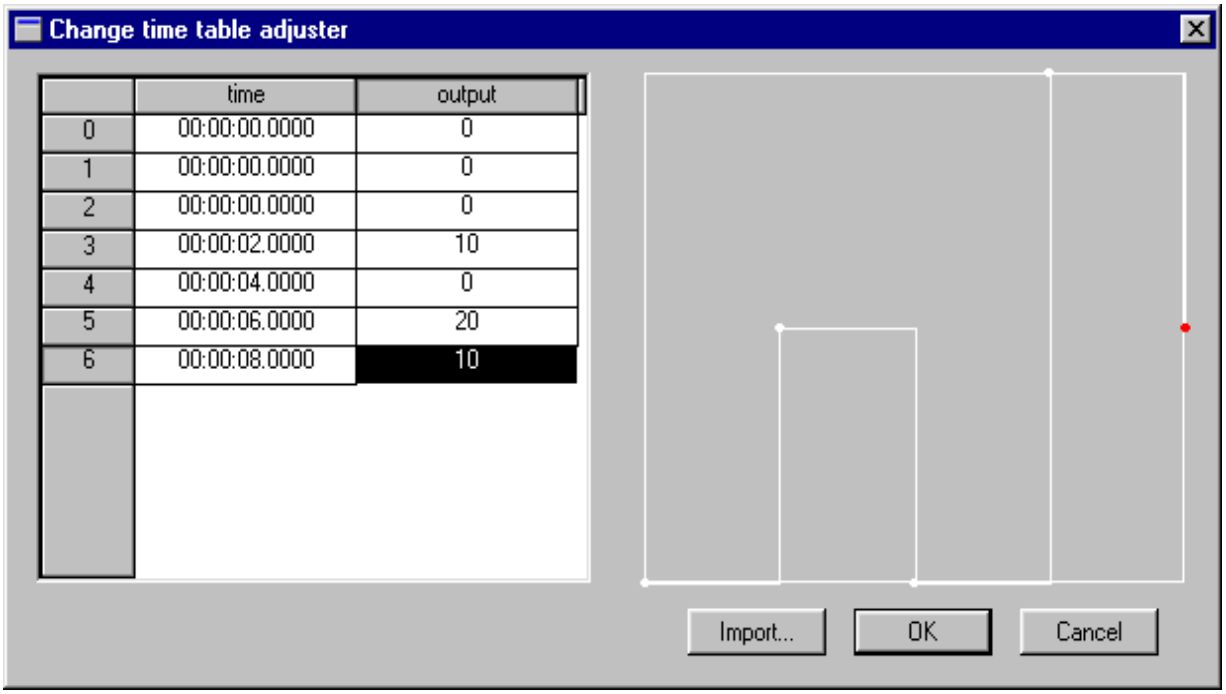
Table 6 Entering and Editing Time Table Values

To change the value of a table cell.	Double-click the entry you want to change, then edit its value in the **Change value** dialog box.
To Insert a new table row of cells.	Press the Insert key on your keyboard, then enter the new time and value.
Import values from an existing file.	Click Import, and choose the file you want to import.

Note: The smallest possible time interval between two entries in the table is 0.0001 seconds. However, make certain that the smallest possible interval that you enter is superior to the value of the base clock timer tick (refer to page 131).

Tip:

Note, in the example shown below, that in addition to being able to view the table of values at left, you can also see a graphical representation of the time-output values in the right-hand area of the dialog box. When you select one of the values in the table, the point in the graph corresponding to the value pair to which the selected value belongs, will turn red.



10.2.6 Setting Sinusoidal Input Signals

If you wish to the input signal to vary sinusoidally, you must attach a **Sine Wave Adjuster** to the appropriate input pin or wire. Sine Wave Adjusters let you set the signal amplitude and frequency that is input to the pin or wire.

Note: You can only attach Sine Wave Adjusters to wires or pins with analog input.

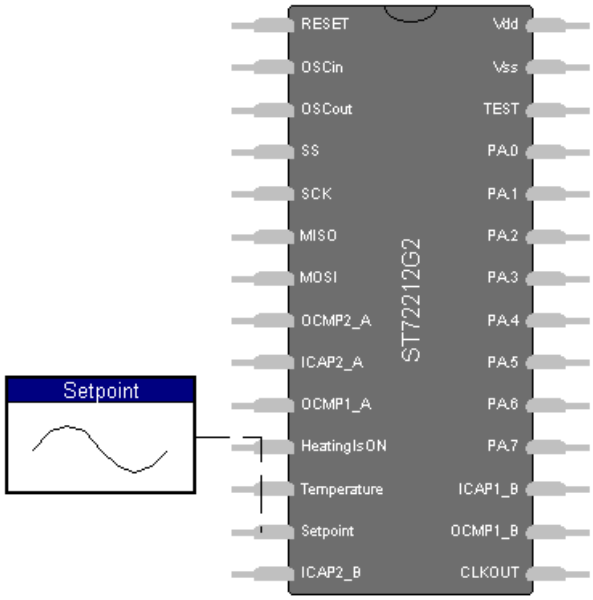
To attach a Sine Wave Adjuster to a pin or wire:

1 Select the pin or wire to which you want to connect the adjuster by clicking on it.

2 Click **.**

3 Click where you want the adjuster to appear.

The adjuster is now placed in your scheme and connected to the pin or wire you selected. In the example at right, we see that the sine wave adjuster has been connected to the pin called “Setpoint”, which is also the name of the input symbol. Note that the sine wave adjuster shows the name of the pin whose input the adjuster controls.

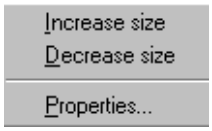


4 Set the signal amplitude and frequency by:

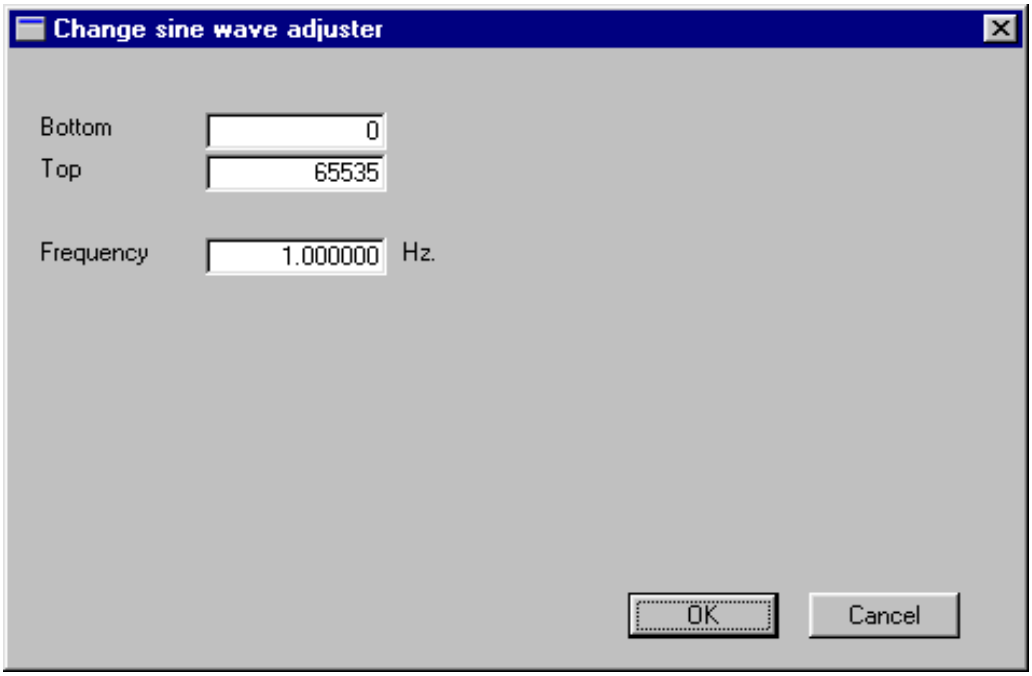
- **Double-clicking the adjuster**,

or,

- **Right-clicking on the adjuster and selecting *Properties* from the popup menu (shown at right).**



The **Change sine wave adjuster** dialog box opens:



5 Fill in the values that will define the amplitude and frequency of the sine wave input:

- To set the bottom range of the wave amplitude, edit the value in the ***Bottom*** field
- To set the top range of the wave amplitude, edit the value in the ***Top*** field.
- To set the wave frequency, edit the value in the ***Frequency*** field.

6 Click OK.

10.2.7 Setting Square Wave Input Signals

To set a square wave input signal, you must attach a **Square Wave Adjuster** to the appropriate input pin or wire. Square Wave Adjusters let you set the signal amplitude, duty cycle and frequency.

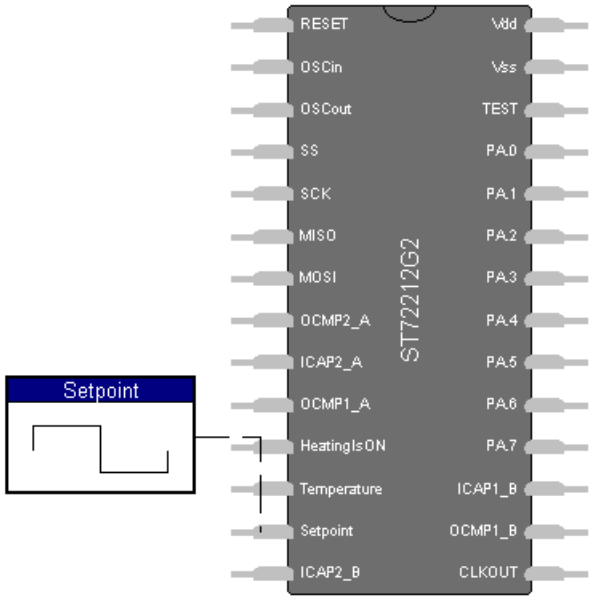
To attach a Square Wave Adjuster to a pin or wire:

1 Select the pin or wire to which you want to connect the adjuster by clicking on it.

2 Click **.**

3 Click where you want the adjuster to appear.

The adjuster is now placed in your scheme and connected to the pin or wire you selected. In the example at right, we see that the square wave adjuster has been connected to the pin called “Setpoint”, which is also the name of the input symbol. Note that the square wave adjuster shows the name of the pin whose input the adjuster controls.

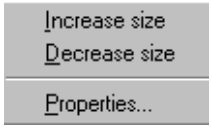


4 Set the signal amplitude, duty cycle and frequency by:

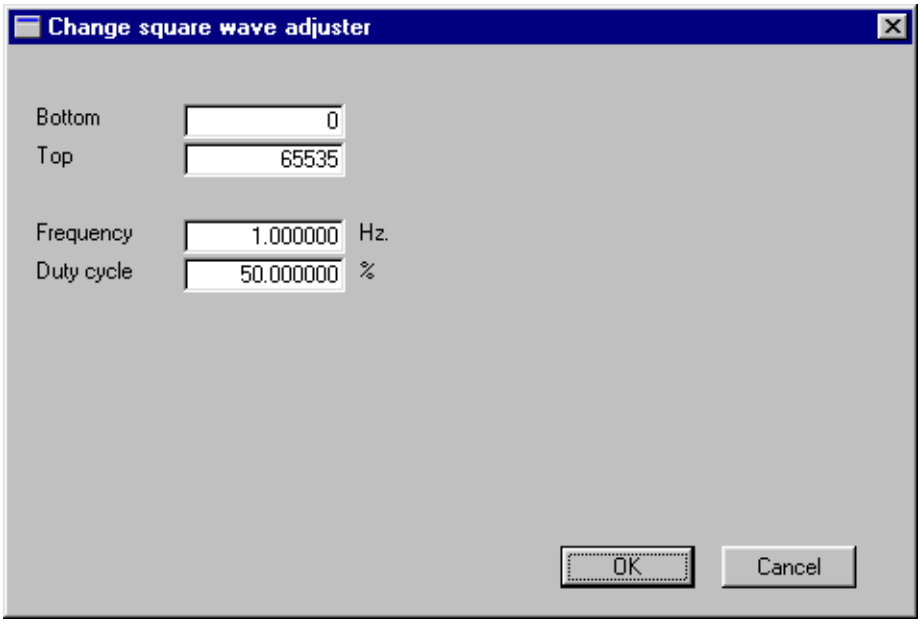
- **Double-clicking the adjuster**,

or,

- **Right-clicking on the adjuster and selecting *Properties* from the popup menu (shown at right).**



The **Change square wave** adjuster dialog box opens:



5 To set the bottom range of the wave amplitude, edit the value in the Bottom field

To set the top range of the wave amplitude, edit the value in the **Top** field.

To set the wave frequency, edit the value in the **Frequency** field.

To set the duty cycle, that is the percentage of time that the signal is at its top value, edit the value in the **Duty cycle** field.

6 Click OK.

10.3 Monitoring Signals with Probes

When you simulate an application, you can monitor the signals it generates using probes. This enables you to experiment with and fine-tune your application.

You connect probes to any wire, pin or symbol whose signals or states you want to monitor.

ST-Realizer lets you connect four types of probes:

- **Numeric Probes**, which let you view the current signal value of the pin or wire in the Binary, Decimal, Hexadecimal and Octal number bases.
- **Oscilloscope Probes**, which let you view the signal value of the pin or wire as a graph.
- **State Machine Probes**, that let you view the value (i.e. current state) of the initial state symbol in a state machine.

The following paragraphs describe how to use probes.

10.3.8 Viewing Signal Values Numerically

To view signal values in numeric form, you must attach a **Numeric Probe** to the appropriate wire or symbol. Numeric Probes let you view the current value of the wire or symbol's signal, in the Binary, Decimal, Hexadecimal or Octal number bases. While the simulation is running, the value that Numeric Probe displays will vary with the changing signal in the wire or symbol.

To attach a Numeric Probe:

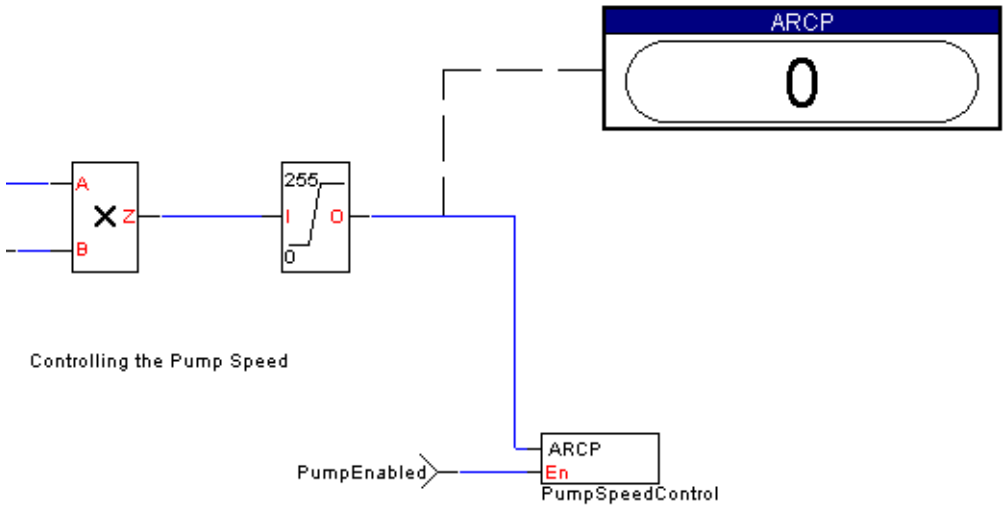
1 **Select the pin or wire to which you want to connect the probe by clicking on it.**

2 **Click** **.**

3 **Click where you want the probe to appear.**

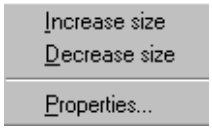
The probe is now placed in your pin-level drawing or scheme and connected to the pin or wire that you selected.

In the example below, we see that the numeric probe has been connected to the a wire in a scheme. Because this wire is an input to a symbol and the symbol pin to which it is connected is called "ARCP", this is also the name of the signal that the numeric probe is measuring. Because of this, the numeric probe also displays the signal name, "ARCP".

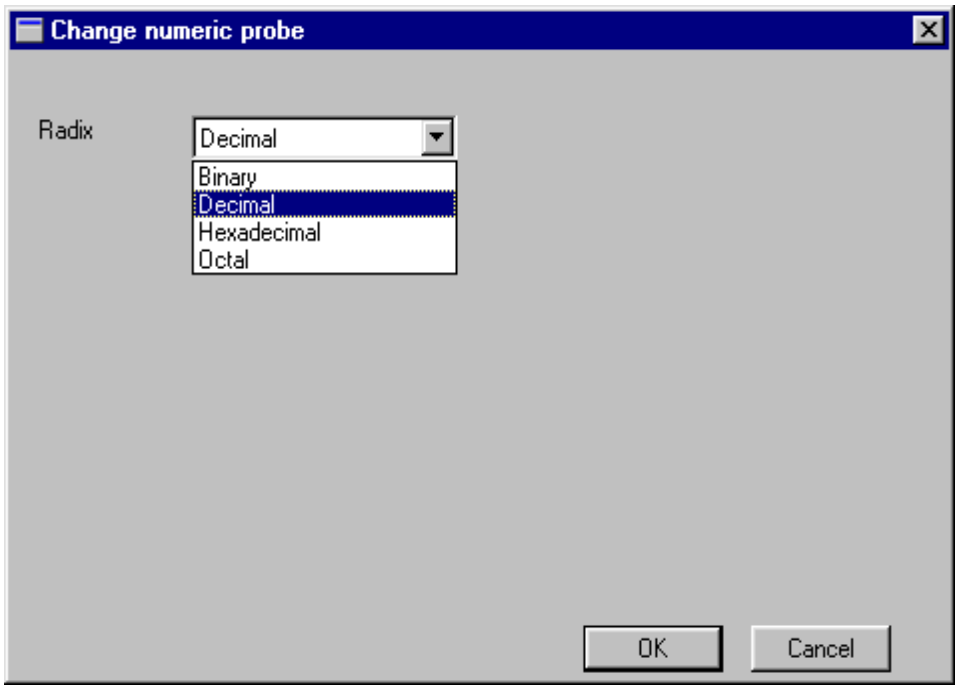


To select the displayed number base:

1 **Double-click the probe, or right-click the probe and select *Properties* from the popup menu (shown at right).**



The Change numeric probe dialog box opens:



2 Select the appropriate number base from the drop-down list, and click OK.

10.3.9 Viewing Signal Values Graphically

To view variable values in graphic form, you must attach an **Oscilloscope Probe** to the appropriate pin or wire. Oscilloscope Probes let you view the value of the pin or wire as a graph. You can also adjust the top and bottom levels of the Y-axis of the displayed graph, and the time scale within which the value is displayed.

To attach an Oscilloscope Probe:

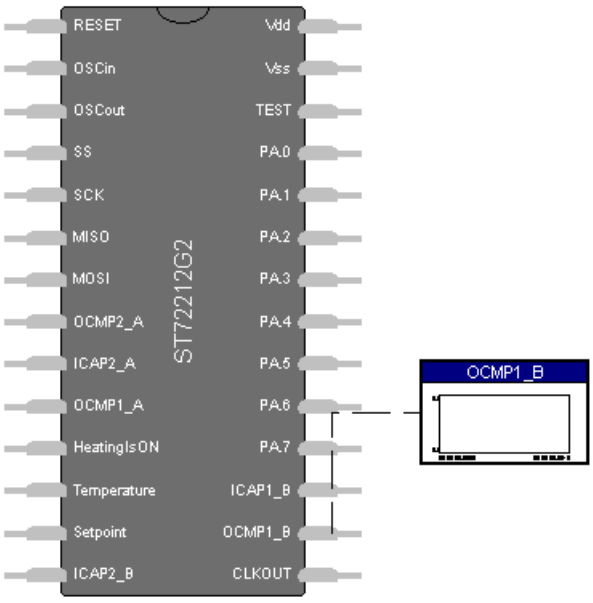
1 Select the pin or wire to which you want to attach the probe by clicking on it

2 Click

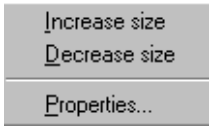
3 Click where you want the probe to appear.

The probe is now placed in your pin-level drawing or scheme and connected to the pin or wire that you selected.

In the example at right, we see that the oscilloscope probe has been connected to the an output pin, called OCMP1_B, in the pin-level drawing. Note that the oscilloscope probe also shows the name of the pin whose output signal the probe is monitoring.



To change the Y-axis, time scale, and time mode of the displayed graph:



1 **Double-click the probe, or right-click the probe and select *Properties* from the popup menu (shown at right).**

The **Change oscilloscope probe** dialog box will open. From this dialog box, you can set options that will control the way in which the signal is displayed. Table 7 explains the various options available in the dialog box.

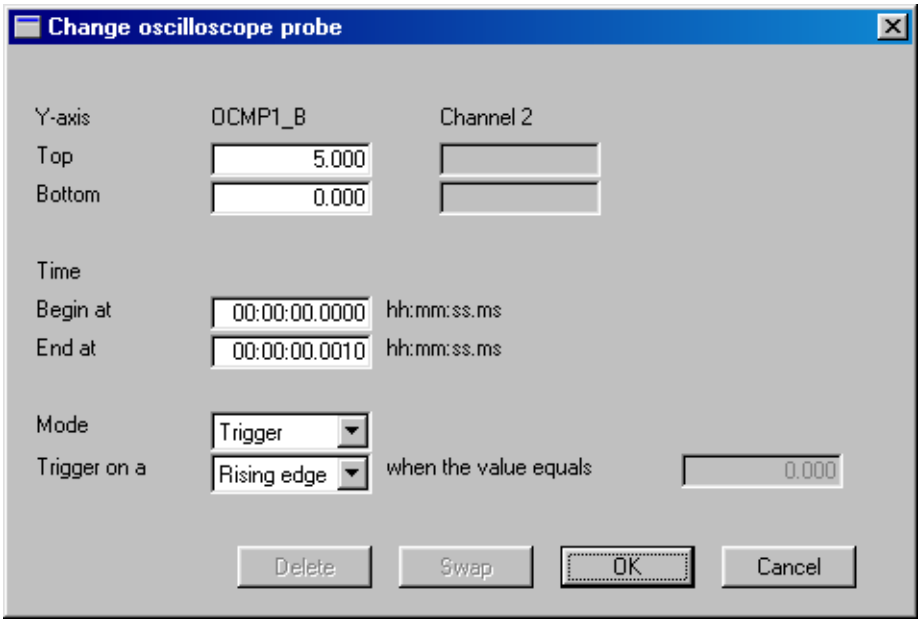
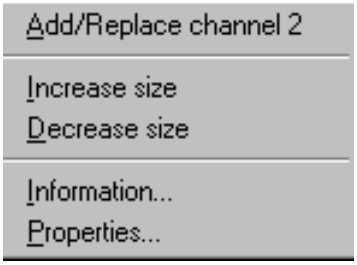


Table 7 Adjusting the Oscilloscope Probe Output

To change the maximum displayed Y-axis value:	Enter the new top Y-axis value in the **Top** field.
To change the minimum displayed Y-axis value:	Enter the new bottom Y-axis value in the **Bottom** field.
To change the time at which the signal display will start (i.e. the beginning of the X-axis):	Enter the new time in the **Begin at** field.
To change the time at which the signal display will end (i.e. the end of the X-axis):	Enter the new time in the **End at** field.
To set the display mode:	The Oscilloscope can display in three modes: • **Single scan mode -** The signal is displayed from the **Begin at** X-value until the **End at** X-value, and then the display picture is frozen. • **Trigger mode -** The signal is continuously displayed such that each time a rising edge, falling edge or a specified value is detected by the probe, the displayed signal restarts at the graph's origin. • **Wrap around mode -** The signal is continuously displayed such that once the X value (time) equals the **End at** value (or multiple of), the displayed signal restarts at the graph's origin.
To add another signal to the second channel:	Right-click on the oscilloscope probe in the simulation view. A popup menu will appear as at right. Select ***Add/Replace channel 2***, and select a second signal (either a wire or a pin) for the second channel.  The oscilloscope probe will now display two signals—one from each channel.

10.3.10 Viewing State Machine States

You can view the states of a state machine by attaching a **State Machine Probe** to the initial state machine symbol. You can choose to see either the current state only or the current state and a set number of previous states.

To attach a State Machine Probe to a state machine:

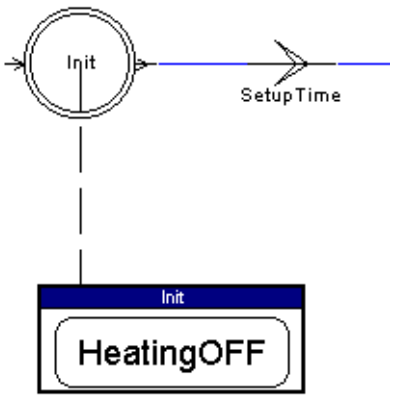
1 Select the initial state symbol to which you want to attach the probe by clicking on it.

Note: You may only attach State Machine probes to initial state symbols in scheme-views of the simulation. State Machine probes cannot be connected to pins, wires or any symbol other than stateinit.

2 Click **.**

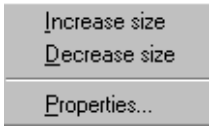
3 Click where you want the probe to appear.

The probe is now placed in the scheme and connected to the state machine you selected.
In the example shown at right, the state machine probe is connected to a *stateinit* symbol in a state machine diagram. The symbol name is called Init, hence the name of the probe is also Init. The value of the probe corresponds to the value of the state which is currently active (in our example, the active state is called HeatingOFF).

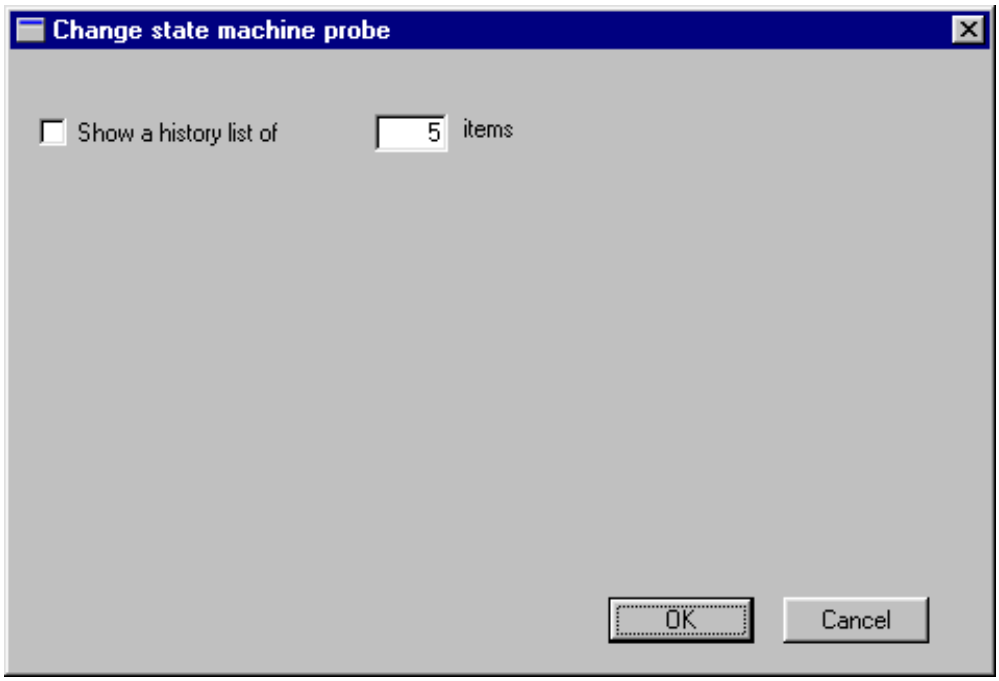


To enable/disable the state history list and set its length:

1 Double-click the probe, or right-click the probe and select *Properties* from the popup menu (shown at right).



The **Change state machine probe** dialog box opens:

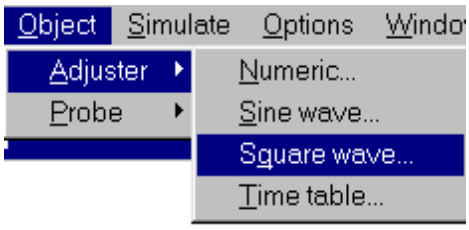


To change the length of the history list, enter a new length in the ***Show a history list of*** field.

2 Click OK.

10.4 Selecting Adjusters and Probes

Instead of using the buttons in the tool bar, you may select any adjuster or probe via the **Object** menu. Just click **Adjuster,** or **Probe**, and select the appropriate object.



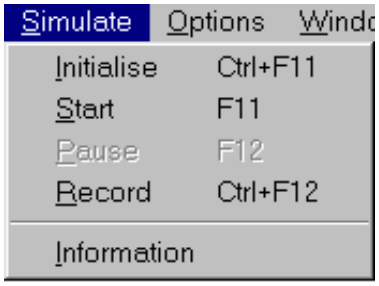
10.5 Running the Simulator

When you run the simulator, the adjusters you placed generate the input values, your application performs its processing, and where appropriate, the results are displayed by the probes you placed.

While the simulator is running, the simulation time is displayed in the status bar at the bottom of the screen.

10.5.11 Starting/Stopping the Simulation

You can initialize, start, or stop the simulation process and proceed to the recording or displaying of information about this process:

To initialize the simulator:	If you have already run the simulator, and you want to **initialize** it to its original state: Click , or: In the **Simulate** menu; click **Initialize**: 
To run the simulator:	Click , or: In the **Simulate** menu; click **Start**.
To temporarily stop your simulation:	Click , or: In the **Simulate** menu; click **Pause**.
To record simulation information:	1 Open a .log file (see “Recording and Reusing Adjuster and Probe Values” on page 160) 2 Under the **Simulate** menu, click ***Record***, or click 3 The red circle changes to a red square:
To display information about the current simulation session:	Under the **Simulate** menu, click ***Information***.

10.5.12 Setting Run Options

Before you run your simulation, you should consider which run options you want to set.

The run options are:

- Whether you want to run the simulation continuously, for a set number of loops, or for a set time period.

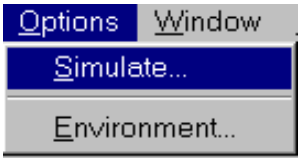
When continuous simulation is chosen, you stop the application when you want. Periodic adjusters such as square wave adjusters and sine wave adjusters are particularly adapted to this mode.

When a set number of loops is chosen, you can execute one loop at a time. Note that in this case a loop time is the time needed to execute all the macro-instructions and not the tick which is the time base used by ST-Realizer.

- Whether you want to use your PC's clock or the microcontroller clock for the simulation. Note that if you use the PC's clock, the sampling time is the end of the execution loop. The corresponding absolute time is recalculated using the PC's clock. This mode is much faster to run since the number of sampled points is lower, however the lower sampling frequency causes a loss of precision.

To set these options:

1 On the Options menu, click *Simulate.*



The ***Simulating*** daughter dialog box opens:

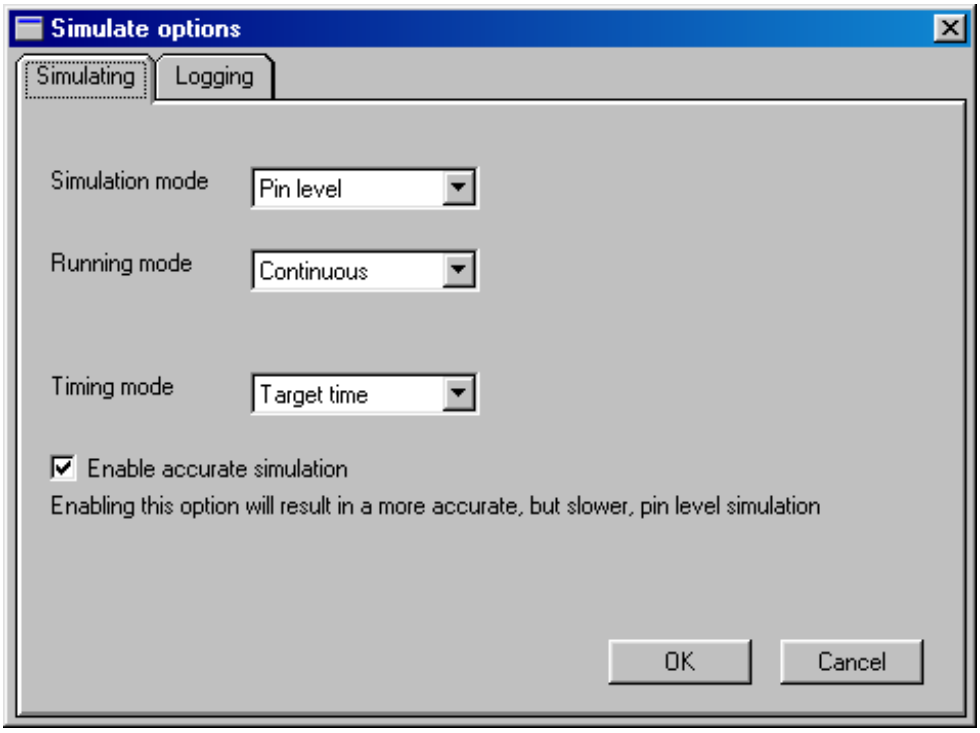


Table 8 Changing the Simulating options

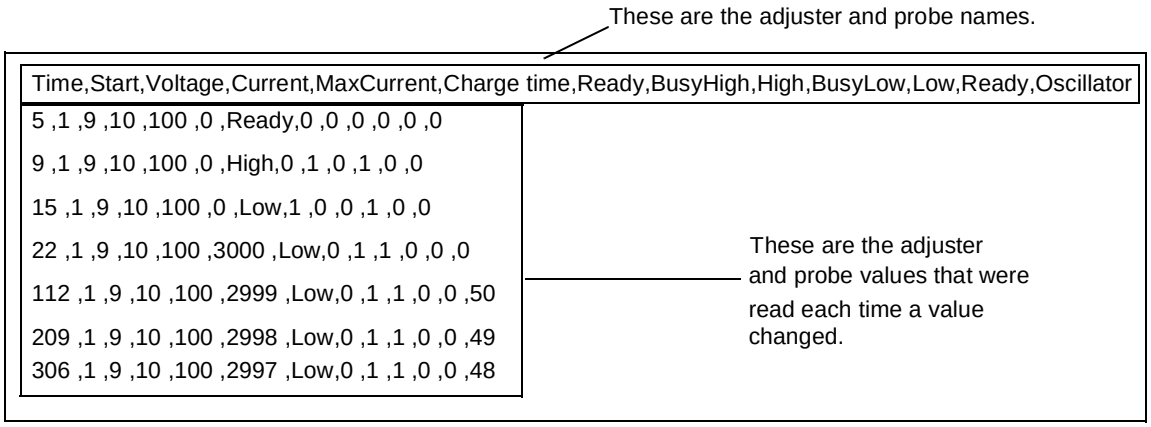
To change the Simulation mode:	There are two simulation modes available: • **Pin level -** This is the default setting. A pin-level drawing of the simulation will be available, as well as all of the application schemes. • **Functional -** Only the scheme-views of the application will be available.
To change the Running mode:	There are three running modes available: • **Continuous -** The simulation will run continuously until you stop or pause manually. • **Limited loops -** You can enter the number of loops (processing cycles) you wish the simulation to run for in the **Stop after** field. • **Limited time -** You can run the simulation for a specific period of time by entering the amount of time in the **Stop after** field.
To change the Timing mode:	There are two timing mode options: • **Use target time -** This option uses the microcontroller clock to time the simulation. • **Use host time -** This option uses the PC's clock to time the simulation.

Note: If you choose to use the microcontroller clock for the simulation, the simulation time is controlled by the ST device, and a scaling effect takes place due to the simulation process. This means that, for instance, the PC may need 10 seconds to represent, say, 100 ms in the simulated application environment. This is the most accurate way of simulating the ST device. If you choose to use your PC clock for the simulation, a time of 10 seconds will take 10 seconds. This mode is less accurate than the target time mode.

10.6 Recording and Reusing Adjuster and Probe Values

You can record the values generated by adjusters and read by probes while a simulation is being run. ST-Realizer records this information in Log files. This information can be useful for viewing the exact adjuster and probe values at any given time during the simulation. You can view log file information by opening the log file in any word processor application.

The format of log files is as follows:


These are the adjuster and probe names.
Time,Start,Voltage,Current,MaxCurrent,Charge time,Ready,BusyHigh,High,BusyLow,Low,Ready,Oscillator
5 ,1 ,9 ,10 ,100 ,0 ,Ready,0 ,0 ,0 ,0 ,0 ,0
9 ,1 ,9 ,10 ,100 ,0 ,High,0 ,1 ,0 ,1 ,0 ,0
15 ,1 ,9 ,10 ,100 ,0 ,Low,1 ,0 ,0 ,1 ,0 ,0
22 ,1 ,9 ,10 ,100 ,3000 ,Low,0 ,1 ,1 ,0 ,0 ,0
112 ,1 ,9 ,10 ,100 ,2999 ,Low,0 ,1 ,1 ,0 ,0 ,50
209 ,1 ,9 ,10 ,100 ,2998 ,Low,0 ,1 ,1 ,0 ,0 ,49
306 ,1 ,9 ,10 ,100 ,2997 ,Low,0 ,1 ,1 ,0 ,0 ,48
These are the adjuster and probe values that were read each time a value changed.

You can also reuse recorded adjuster values in subsequent simulations in order to recreate identical conditions.

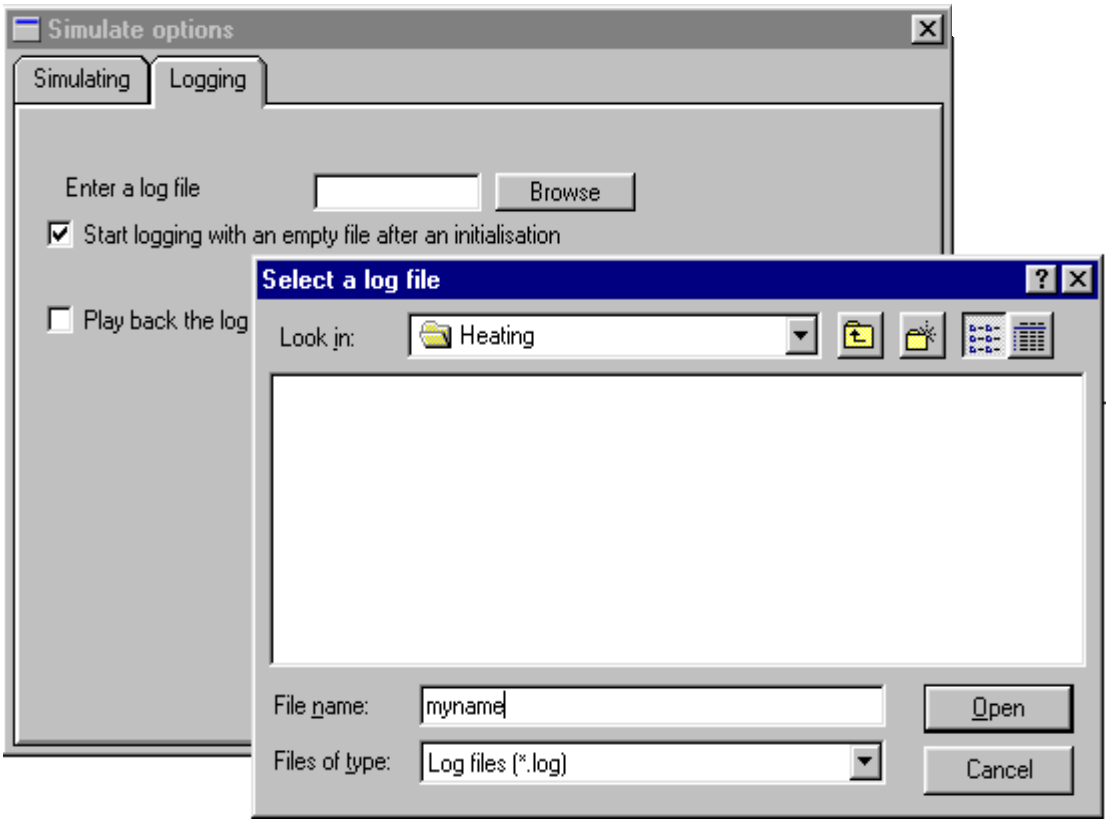
10.6.13 Recording Adjuster and Probe Values

1 Before running the simulation whose values you want to record, on the Options menu click Simulate.

The Simulate Options dialog box opens.

2 Click the Logging tab.

The ***Logging*** daughter dialog box opens.



3 Type the name of the log file you want to create in the “start logging...” box, or browse through the appropriate folder.

4 Click Open.

5 Run your simulation. The log file will be recorded while your simulation is running.

6 Once you have finished recording simulation information, close the log file by clicking **.**

10.6.14 Reusing Adjuster Values

Once you have recorded a log file, you can reuse the recorded adjuster values in subsequent simulations:

In the Simulate Options dialog box:

1 Click the Logging tab.

The Logging daughter dialog box opens as shown on page 161.

2 Type the name of the log file you want to create in the “play back...” box, or browse through the appropriate folder.

3 Click Open.

4 Run your simulation. The adjuster values recorded in the log file are input to your simulation while it is running. Note that you can also manually change adjuster values.

5 Once you have finished recording simulation information, close the log file by clicking .

11 CREATING YOUR OWN SYMBOL

11.1 Overview

You create or edit your own symbols using ***ST-Symbol Editor***. Creating a symbol involves the following tasks:

- Creating a symbol shape.
- Placing pins that represent the variables that are input to and output from your symbol.
- Linking your symbol to the macro(s) it represents.

Once you've finished creating your symbol, you can create a customized library containing all the symbols that you used in a project, including symbols that you created yourself.

The creation of new symbols falls into two categories:

- **The creation of new subscheme symbols.** These are the symbols that represent subschemes in the root scheme. A number of subscheme symbols are included in the main library, but you may find that you need to create your own because you require a different number of input and output pins that available in those symbols provided.
- **The creation of new user-defined symbols.** This is a means by which you can design your own symbol to perform a customized function. **Note that creation of a new user-defined symbol requires a sound understanding of how to program in assembler code as it will be necessary to write the macros which define how your symbols work.**

The creation of both of these types of symbols is described in this chapter. In both cases, you use the ST-Symbol Editor to create your symbol.

Note: The ST-Realizer online help system includes a tutorial which leads you through the steps of creating an example symbol and its macros.

11.2 Running the ST Symbol Editor

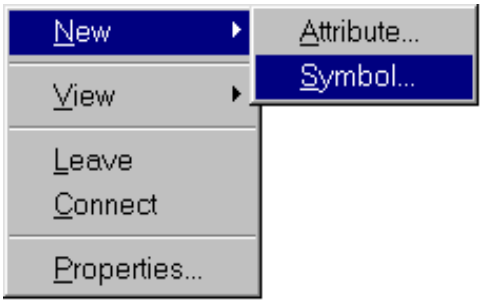
To run ST-Symbol Editor from the ST-Realizer window:

1 **Make sure you have a scheme loaded in the ST-Realizer window**

2 **On the Tools menu, click Symbol Editor**

or:

Click the scheme area out of any symbol with the right mouse button and select the New→Symbol... menu sequence.



The ST-Symbol Editor window opens.

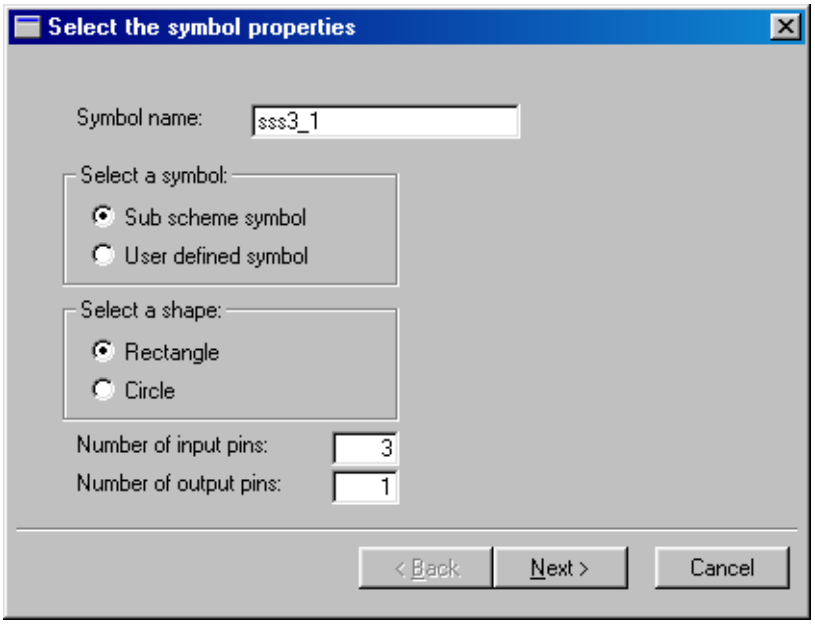
It automatically shows the symbol editor's tool bar and the **Select the Symbol properties** dialog box:

You can now define either a new subscheme symbol or a new user-defined symbol.

11.3 Defining a New Subscheme Symbol

Subscheme symbols are, effectively, single-symbol representations of an entire subscheme, that can be used to reference a subscheme from within another scheme. Some subscheme symbols have already been defined in the ST-Realizer library, but you may find that you need to create a subscheme with, say 3 inputs and only one output. We will use this example to help explain how to easily create a new subscheme symbol.

1 Open the Symbol Editor.



2 Give the new subscheme symbol a name.

An ***"sssx_y"*** type name is recommended, to be consistent with standard subscheme names that can be found in the main symbol library. An ***"sssx_y"*** subscheme has ***x*** inputs and ***y*** outputs.

3 Click the “Sub scheme symbol” radio button.

4 Select the shape you which to assign to the symbol.

The shape of the symbol can be either rectangular or circular.

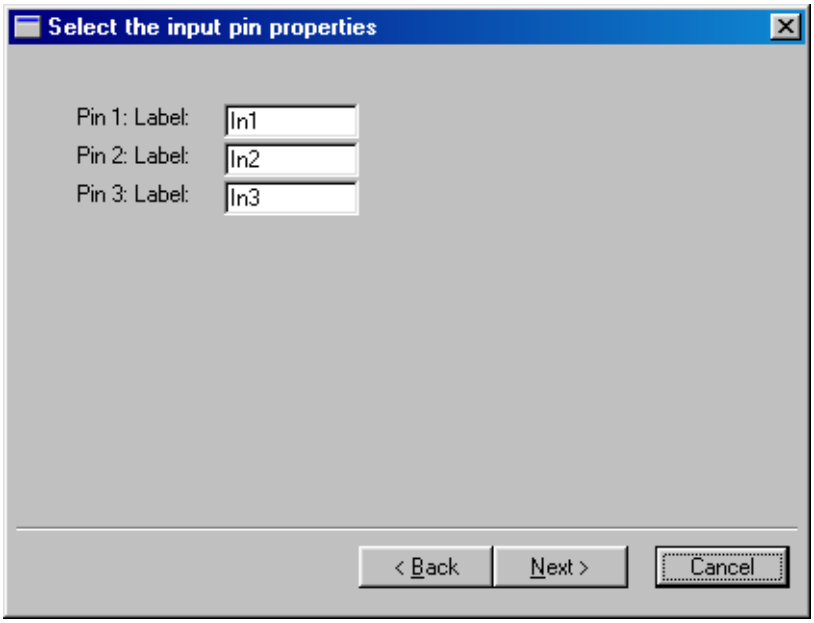
Note: The shape of the symbol represents, but does not affect, the process that is behind it. Symbol shapes should not cross the square box in the ST-Symbol Editor window (this represents the physical limits of the symbol).

5 Specify the number of input and output pins.

Tip: *All of the above options can be modified later, using the Symbol Editor.*

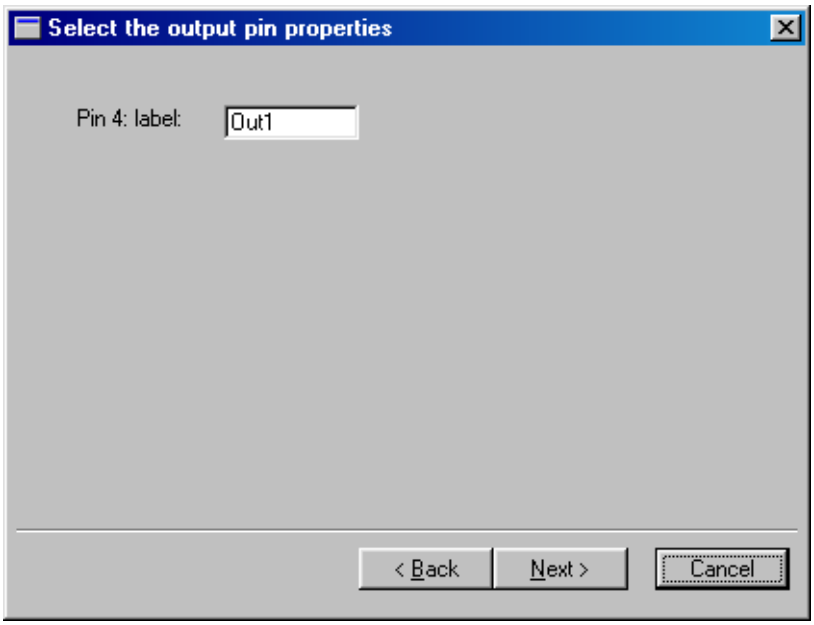
6 Click the Next button.

A new dialog box opens where you specify labels for the subscheme input pins.



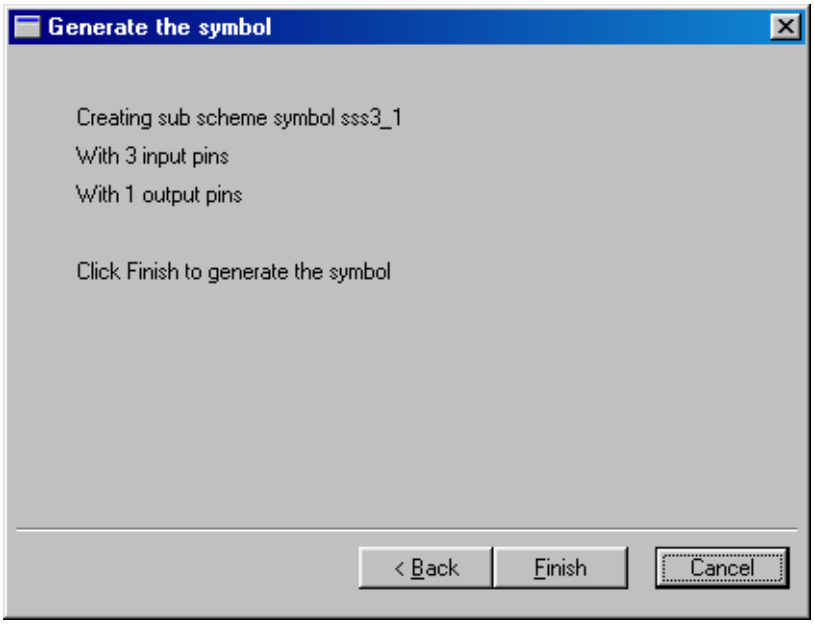
Click **Next** when you’ve finished.

A new dialog box opens where you specify labels for the subscheme output pins.

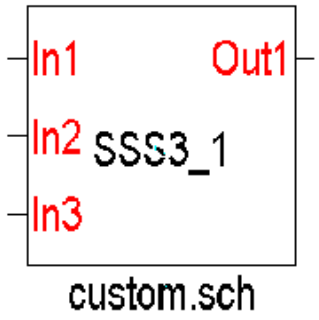


Click **Next** when you've finished.

The **Generate the symbol** dialog box opens.



Click **Finish** to generate the new symbol.



The new subscheme symbol is automatically added to the ***local symbol library*** for the project in which you are working. To save the symbol in a library accessible to all projects, follow the instructions in Section 11.3.1.

To place it in your scheme, click as explained on page 73.

In our example at left, the new subscheme symbol has 3 input pins and one output pin. The associated subscheme is named "**custom.sch**", however, you may choose to leave this attribute field non-specific (by leaving the "?" placed by default, for example).

11.3.1 Adding Your New Subscheme Symbol to a Library

In order to use your new subscheme symbol (or symbols, if you have created more than one) in any ST-Realizer project, you must save it to a library file located in the **`<root folder>\Lib`** folder. To do this:

1 **Open the project where you have created the new subscheme symbol(s). Open the scheme where the new subscheme symbol is.**

2 **Open the local symbol library by clicking** .

3 **Create a new blank scheme in the project by selecting *File→New→Scheme*. Call it any name you wish (for example, *temporary.sch*).**

4 **Using the local symbol library, select and place the new subscheme symbol(s) that you wish to save to a library in the blank scheme.**

5 **Now you must edit the new subscheme symbol(s).**
Right-click the new subscheme symbol and select *Edit* from the popup menu. The symbol editor will open. Double click on the subscheme filename attribute at the bottom of the symbol (i.e. in the above example, we gave the value "custom.sch" to this attribute, but by default it is left as a "?").
At this point you have to make a choice, depending on how you wish the default usage of the symbol to be:

- If you wish to leave the subscheme file generic, leave the "?" in the field. Every time you use the symbol in the future, you will have to remember to fill in this attribute value, specifying the filename of the subscheme (and it's path if it is not located in the project folder). Alternatively, if you double-click the symbol (once placed in a project scheme), you will be prompted to create a new subscheme.

- If you always wish your new subscheme symbol to reference the same subscheme file, enter the name of the subscheme file (for example, custom.sch) in this attribute field with its correct path (i.e. the project folder in which the subscheme file will occur).

Note: Remember that if you enter a filename in this attribute field, the path by default will be the path of the project in which you created/edited the subscheme symbol.

To be able to access a particular subscheme **from any project**, we recommend that you copy the subscheme file you have specified (for example, custom.sch) to the **<root folder>\Lib** folder and, using the symbol editor, in the filename attribute field enter path and filename of the subscheme as follows: **<root folder>\Lib*filename*.sch.** For example, in order to use the subscheme custom.sch in any project, we would place a copy of custom.sch in **<root folder>\Lib** and using the symbol editor, enter **<root folder>\Lib\custom.sch** in the subscheme filename attribute field.

Note: The exact <root folder> path depends on where you installed ST-Realizer on your PC. By default, this path is C:\\Program Files\ST-Realizer\Lib\...

Exit the Symbol Editor, saving your changes.

Repeat the above steps for each new symbol you have included in the scheme.

6 **While in the subscheme you created in Step 3 (i.e. *temporary.sch*), select *File→Save as..* from the main menu and save the file as a library file (.lib extension) in the library folder (<root folder>\Lib). Give the new library a suitable name—for example, my_symbols.lib. Your new subscheme symbols will be stored in this library file, accessible to all projects.**

Tip: *When you save a scheme as a library, you will copy all of the symbols in that scheme to the library. Therefore, to save only new subscheme symbols you have created to a library, they need to be in a scheme by themselves.*

7 **Exit, without saving, the project in which you have created and modified you new subscheme symbols. (If you save the project, all of the modifications you made to your new subscheme symbols to apply to the project you performed them in.)**

Now, each time you wish to use your subscheme symbol(s) in a new project, simply open the library file you created and select and place the symbols as you would normally do from any other symbol library.

11.4 Defining a New User-Defined Symbol

You can use the symbol editor to create your own symbol, using the User-Defined symbol option.

Note: Defining a new user-defined symbol will require programming ability in the assembler language.

There are four steps in creating a new user-defined symbol:

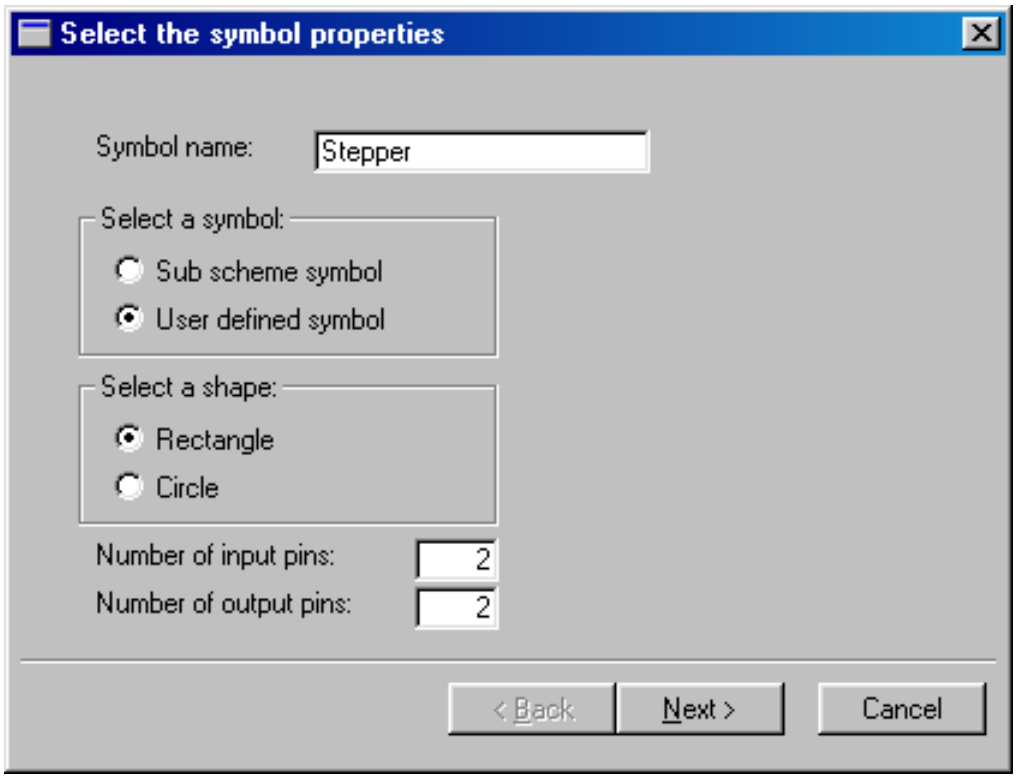
- The definition of the graphic aspect of the symbol, including assigning the required number of input and output pins, naming these pins and assigning pin data types and attributes.
- Editing the new symbol to further refine its appearance.
- The creation of an assembler macro header—essentially the declaration line of the assembler code behind the symbol.
- Finally, writing the assembler macros that define the symbol's function.

The above steps are general, and it is clear that detailed instruction on how to create a new user-defined symbol will vary depending of the symbol function. It is not within the scope of this document to give instruction on how to write assembler code, so in the steps that follow, we will explain, using an example, how to define a new symbol.

The example we will show here is a stepper symbol—a symbol whose function will be to increase an input value by a fixed increment upon detection of a second signal input signal. The stepper symbol will output both the incremented value and its complement.

11.4.1 Defining the New Symbol

1 Open the Symbol Editor.



2 Give the new symbol a name.

In our example, above, we have named our new symbol “Stepper”.

3 Click the “user-defined symbol” radio button.

4 Select the shape you which to assign to the symbol.

The shape of the symbol can be either rectangular or circular. For our example, we will choose a rectangle shape.

Note: The shape of the symbol represents, but does not affect, the process that is behind it. Symbol shapes should not cross the square box in the ST-Symbol Editor window (this represents the physical limits of the symbol).

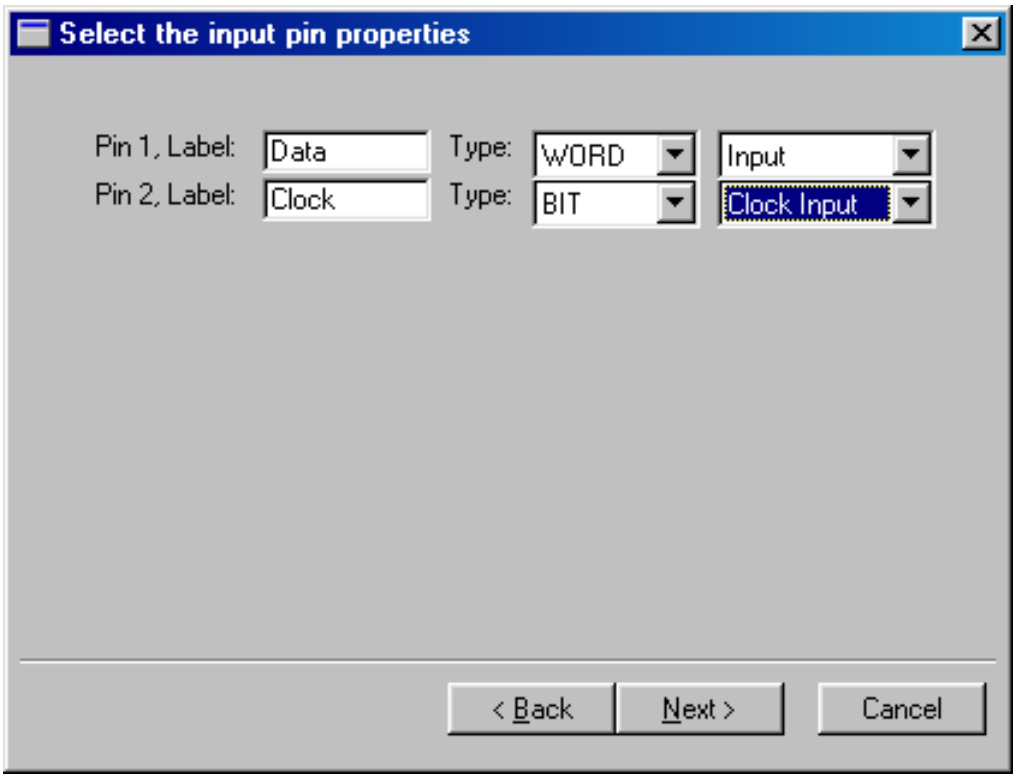
5 Specify the number of input and output pins.

For the stepper symbol, we require two inputs (one for the initial value to be incremented and one for the clock that will control the frequency of the incremental steps), and two outputs (one for the incremented value and the second for the complement of the incremented value).

Tip: *All of the above options can be modified later, using the Symbol Editor.*

6 Click the *Next* button.

7 A new dialog box opens for you to specify the input pin data types:



There are a number of data types to choose from: BIT, UBYTE, SBYTE, UINT, SINT, LONG and WORD. In addition, you can specify a sub-type—either Input or Clock Input.

For a pin transmitting this value range:	Choose this data type:
0 or 1	Bit (BIT)
0 to 255	Unsigned byte (UBYTE)
-128 to 127.	Signed byte (SBYTE)
0 to 65535.	Unsigned int (UINT)
-32768 to 32767.	Signed int (SINT)
-2147483648 to 2147483647.	Long (LONG)
Any values except BIT-type values	Word (WORD)
Any	No type

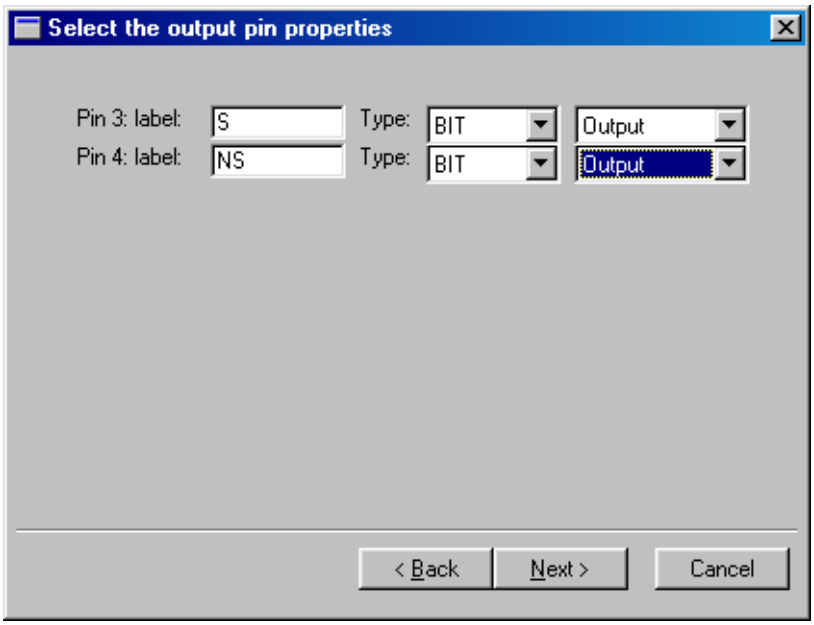
For a detailed description of data types and attributes, refer to Appendix A: “Variables and Attributes” on page 203.

For our example, we will assign the following properties:

- For input pin 1, label = “Data” and type = WORD/Input.
- For input pin 2, label = “Clock” and type = BIT/Clock Input.

Click *Next*.

8 A dialog box opens for you to specify the output pin data types.



Choose data types for each of the output pins.

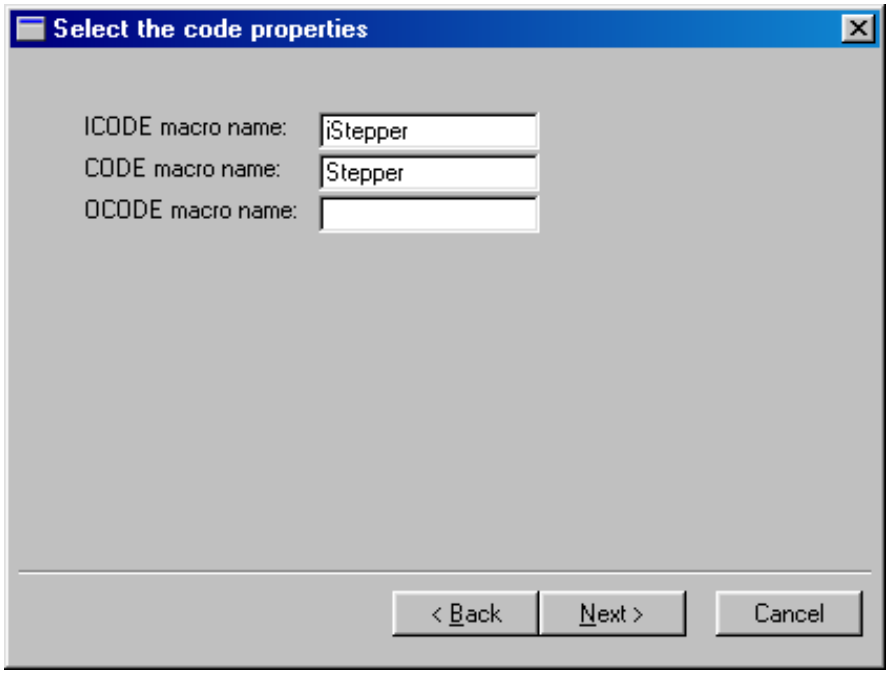
In our example, the data type we wish to assign (Max1) doesn’t appear in the pull-down menu, which only contains the most common data types.

For the time being, assign any data type, for example:

- For output pin 3, label = “S” and type = BIT/Output.
- For output pin 4, label = “NS” and type = BIT/Output.

We will modify the data types on page 178, once the symbol has been created. **Click *Next*.**

9 A new dialog box opens to prompt you for the code properties of the new symbol.



Use this dialog box to link your symbol to the macro(s) it represents.

You can link three macros to a symbol, each with a different execution time:

- At initialization.
- As part of the main loop.
- At the end of the main loop.

Macros that are linked to ST-Realizer symbols must have headings that are in a specific format:

To link a macro that's executed at initialization:

- In the **ICODE macro name** field, enter **i***<macroname>* where *<macroname>* is the name of the macro. For our example, the ICODE macro name is "iStepper".

To link a macro that's executed as part of the main loop:

- In the **CODE macro name** field, enter *<macroname>* where *<macroname>* is the name of the macro. For our example, the CODE macro name is "Stepper".

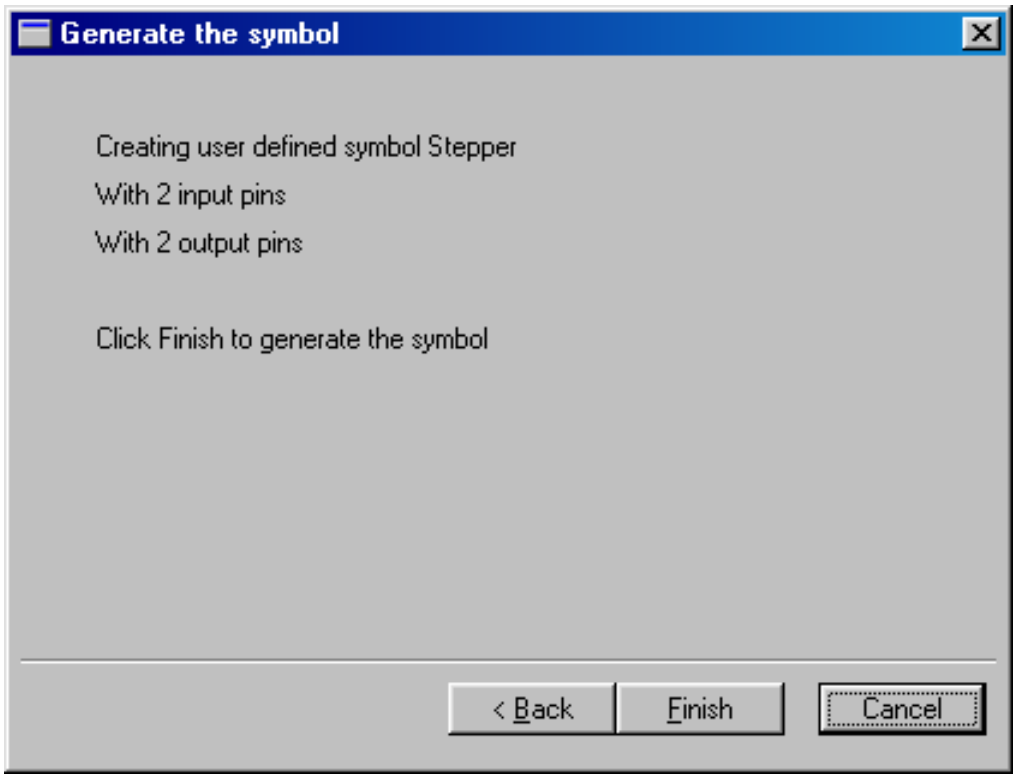
To link a macro that's executed at the end of the main loop:

- In the **OCODE macro name** field, enter **o***<macroname>* where *<macroname>* is the name of the macro. For our example, we do not wish to generate an OCODE macro.

Note: Recall that names in ST-Realizer are case-sensitive.

Click **Next** to continue.

10 The Generate the symbol dialog box opens. Click Finish to confirm.



The new symbol will appear in the scheme, similarly to our example symbol shown at left. We can see our two 2 input pins and two output pins.
The next step is to edit the symbol in order to give the pins names that are more descriptive, to change the pin data types if necessary, and to add some graphical detail to better distinguish the symbol.

11.4.2 Editing the New Symbol

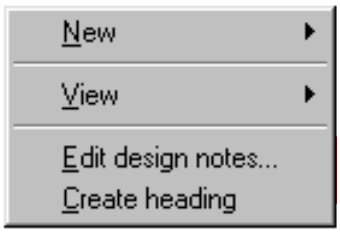
Once you have generated the new symbol, it will appear as above in a symbol editor window that is called by the name of the new symbol. In this environment, you can further edit the visual appearance of the symbol, and refine its attributes.

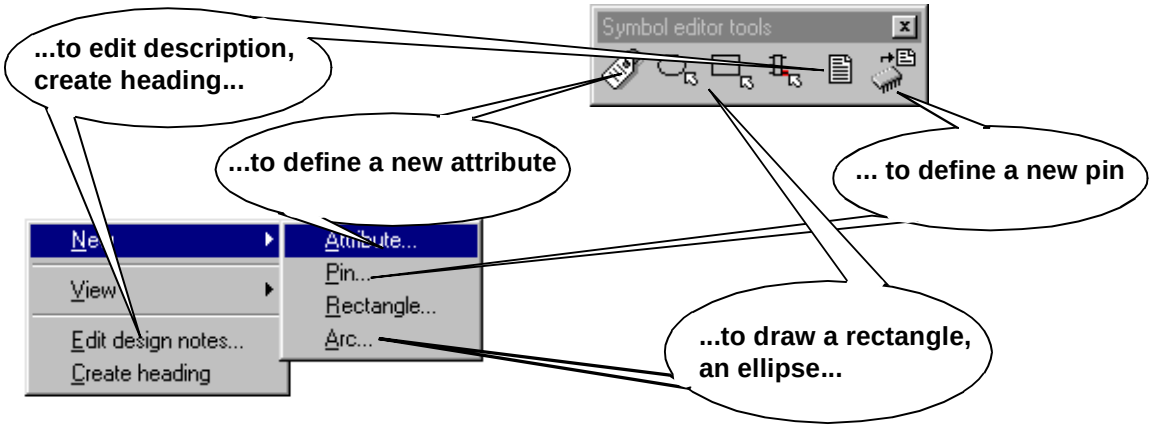
11.4.2.1 Accessing Editing options from the Symbol Editor window:

- In the scheme, right-click the symbol you wish to edit.

  The symbol editor popup menu will appear as at right, allowing a number of options. Most of these options also appear in a toolbar which floats in the symbol editor window.

  The figure below shows how you can use either the popup menu or the toolbar to edit the symbol.



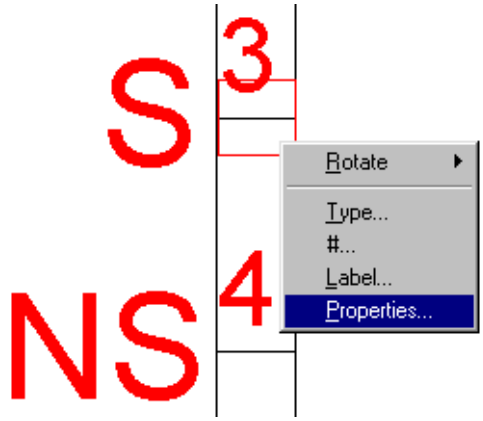


11.4.2.2 Editing pin attributes:

1 Select the pin whose attributes you wish to modify.

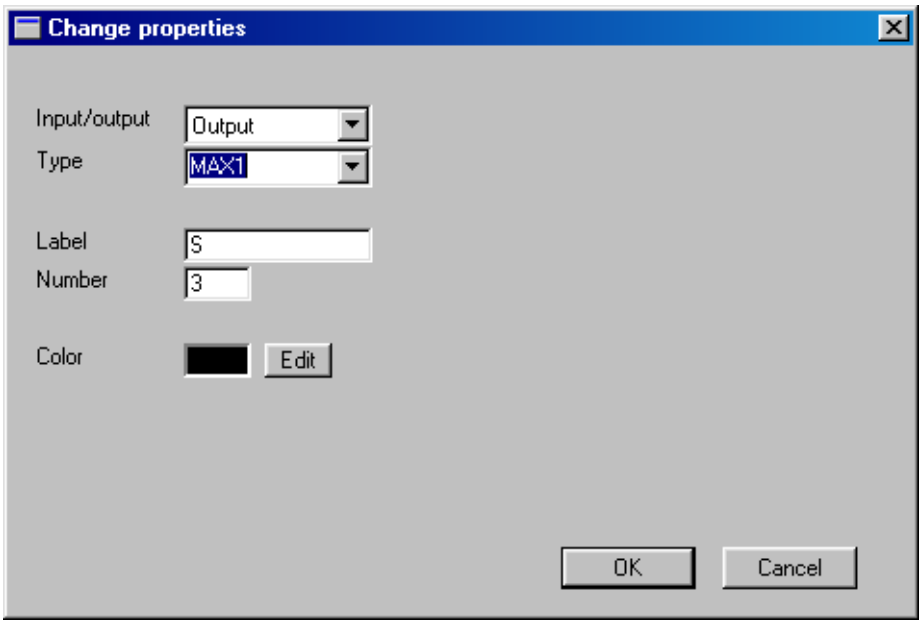
2 Right-click the mouse with the pin selected.

A popup menu will appear as below.



3 Select *Properties* from the popup menu.

A **Change properties** dialog box, like the one shown below, will open.



In our example, we will change the pin 4 (“S”) data type to MAX1 (this option takes the data type of input pin 1. (The data types available in the cascading menu are the most widely used, but they not the only data types available. For a complete list of data types, refer to Appendix A: “Variables and Attributes” on page 203.)

Perform the same operation to the NS pin (pin 4), setting the data type to Max1 as well.

When finished, your symbol should now look as pictured at right.

11.4.2.3 Adding graphical detail to symbols

You can add simple graphics to symbols to help identify them. Basic drawing functions are available from the popup menu or from the symbol editor toolbar (refer to page 177).

To draw an ellipse (or a circle):

1 Click on the symbol editor toolbar, or select New→Arc in the popup menu.

2 Click once where you want the centre of the ellipse to be.

3 Move the cursor in any direction. An ellipse shape appears that is based on the relationship between where you clicked and the current cursor position.

Tip: *Note that the position of your cursor, in x and y coordinates, appears in the status bar at the bottom left of the symbol editor window, whenever you are in drawing mode.*

4 Click when the ellipse is the size and shape you desire.

The ellipse is now drawn. The status bar displays the ellipse's measurements.

To draw a rectangle:

1 Click in the symbol editor toolbar, or select *New→Rectangle* in the popup menu.

2 Click where you wish to place the top left corner of the rectangle.

3 Move the cursor downwards and to the right. Click when the rectangle is the size and shape you desire.

The rectangle is now drawn. The status bar displays the rectangle's measurements.

To draw lines:

1 Click in the main toolbar.

2 Click where you want the line to start.

3 Click where you want the line to end. You can continue drawing connected lines by placing the cursor where you want the next line to end then clicking.

4 Right-click when you want to stop drawing connected lines.

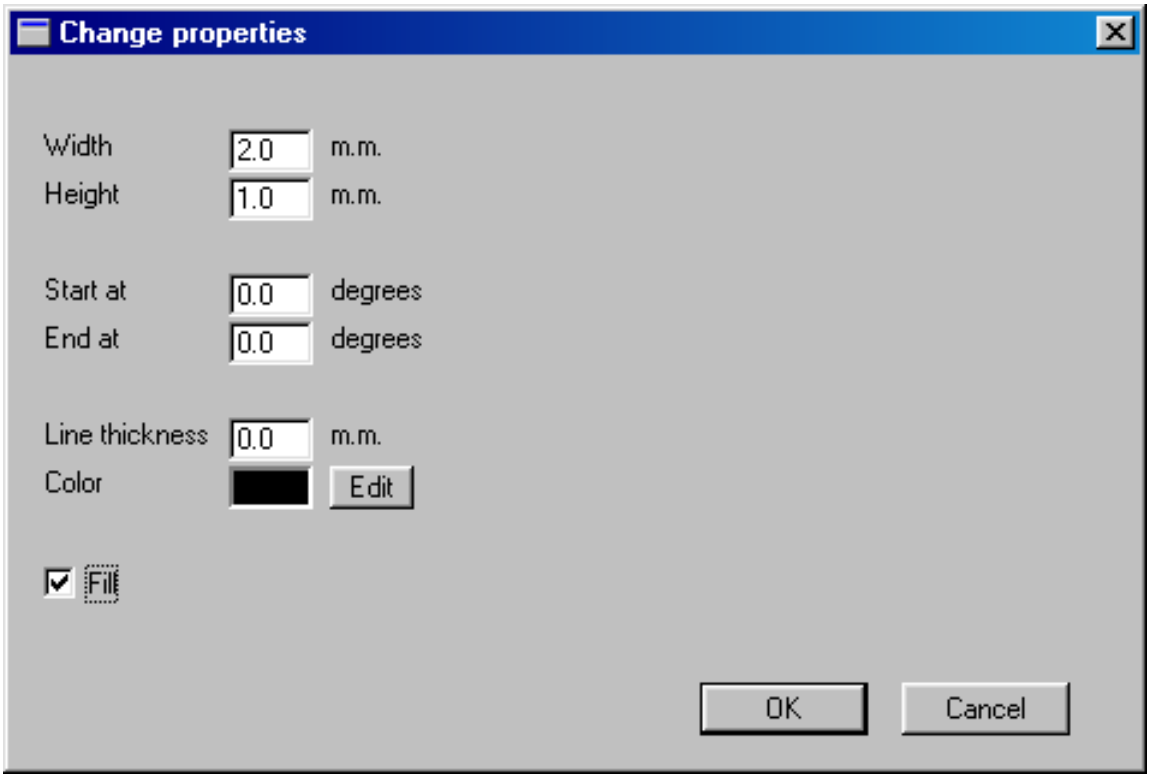
Note: When you draw the cursor automatically snaps to an invisible grid. If you want to change the spacing between the gridlines, on the **Options-Environment** menu, click the **Symbol** tab, then specify the appropriate **grid** value.

Changing graphic properties:

1 Select the graphical object you wish to modify by clicking once on it.

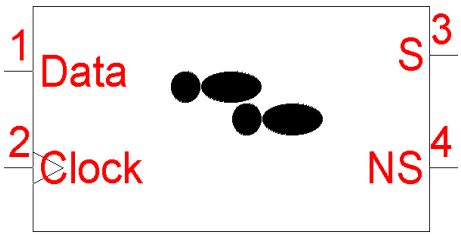
2 Right-click the mouse. A popup menu appears with one options: *Properties*. Select it.

3 A dialog box entitled Change properties opens.



In this dialog box, you can change the dimensions of the graphic, the angle (if the graphic is a line), the line thickness and color, and finally, you can choose to fill the graphic.

In our Stepper example, we are going to add a small graphic to help identify the symbol:



- In the symbol editor window, select from the symbol editor toolbar, or select ***New→Arc*** in the popup menu.
- Using the ellipse drawing tool, draw the footsteps as shown.
- Using Change properties dialog box, fill the four ellipses that make up the footsteps.

11.4.2.4 Symbol Information

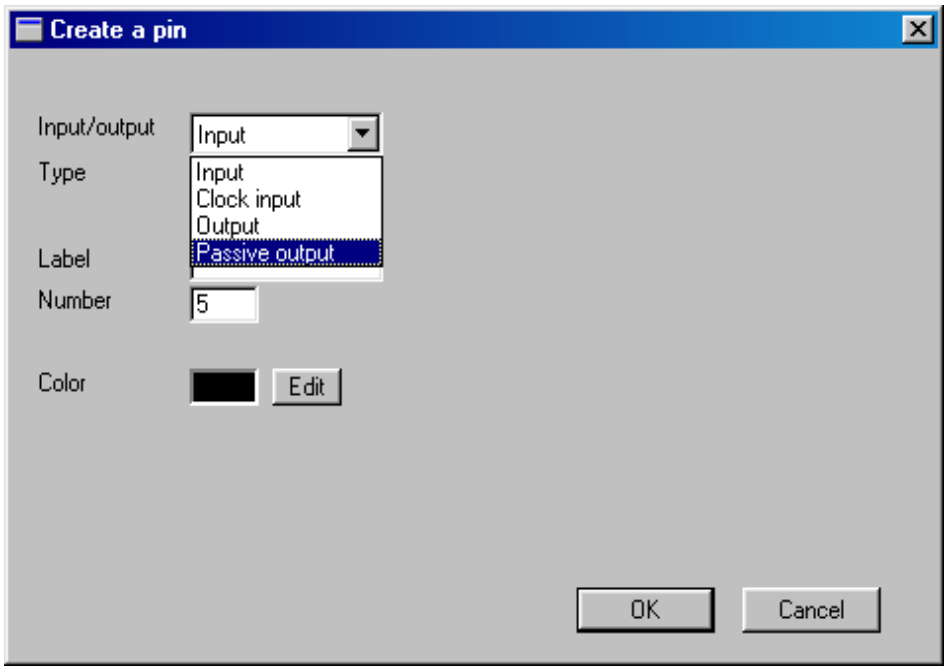
To change the text that describes the symbol (when you click the button) select ***Edit design notes*** in the popup menu and enter the description you wish in the dialog box.

11.4.3 Adding Pins to Your Symbol

The pins you place on your symbol represent the variables that are input to and output from the symbol you are defining.

To place a pin:

1 Click **in the symbol editor toolbar, or select *New→Pin* in the popup menu.**



The Create a pin dialog box opens.

2 The pin number is automatically generated and displayed in the Number field. This is reflected in the structure of the arguments for the assembly macros. The pin number sets the sequence of parameters.

3 In the Label field, enter a label for the pin. This corresponds to the variable name.

4 In the Input/output field, select the type of pin you want to place:

Pin Type	Description
Input	Normal input pin.
Clock input	Input pin that can distinguish between a rising edge, a falling edge and a plateau, by storing the previous input value and comparing it to the present input value.
Output	Normal output pin. There can only be one output pin on a net.
Passive output	Passive output pin. An example of a passive output is a state. There can be more than one passive output pin on a net.

5 In the Type field, select the capacity of the pin you want to place:

For a pin transmitting this value range:	Choose this data type:
0 or 1	Bit (BIT)
0 to 255	Unsigned byte (UBYTE)
-128 to 127.	Signed byte (SBYTE)
0 to 65535.	Unsigned int (UINT)
-32768 to 32767.	Signed int (SINT)
-2147483648 to 2147483647.	Long (LONG)
Any values except BIT-type values	Word (WORD)
Any	No type

6 If you want to change the pin color, click the *Color* box, then choose a new color from the palette.

7 Click OK.

8 A rectangle now appears, representing the position of the pin. Note that it starts at the left side of the symbol boundary.

If you want it to start at another side of the symbol boundary, click . Each time you click this button the pin moves anti-clockwise to the next boundary.

9 Move the cursor to the point of your symbol to which you want to connect the pin, then click to place the pin.

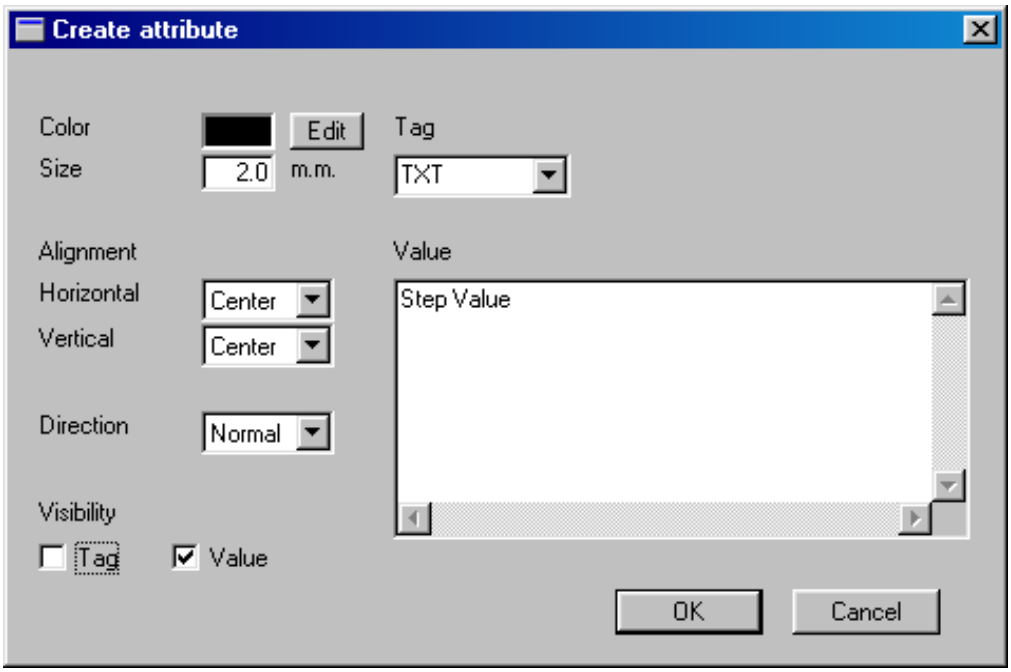
11.4.4 Assigning Attributes to Your Symbol

Attributes allow you to attach extra characteristics to symbols. For example, attaching an attribute of type DEVICE to a pin extends the macro that is linked with the symbol with additional information. The available attribute types are listed in Appendix A: “Variables and Attributes” on page 203.

To assign an attribute:

1 **Click** **in the symbol editor toolbar, or select *New→Attribute* popup menu.**

The Create attribute dialog box opens.



2 **In the *Tag* field, select an attribute tag from the drop-down list.**

For example, for our Stepper symbol, we'd like to add some text. Select the TXT tag from the cascading menu.

3 **In the *Value* field, enter the tag value.**

For our Stepper symbol, we'd like to enter the text "Step Value".

4 **If you want the tag or the value to be hidden, click the appropriate check box in the Visibility box.**

When the boxes are checked, the tag or values are visible. Hidden tags and values can be made visible by choosing *View→Invisible attributes* from the main menu.

For our Stepper symbol, we only wish to view the text, and therefore only check the Value box.

5 **To change the attribute color, click the *Color* field and choose a new color.**

6 **To change the attribute size, edit the value in the *Size* field.**

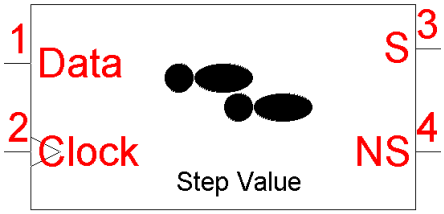
7 **If you want to change the vertical and horizontal alignment, select the *Alignment* type from in the *Vertical* and *Horizontal* drop-down lists.**

8 **Click OK when you have finished defining the attribute.**

9 **The cursor becomes a ghost box indicating the size of the attribute. Position the cursor over the part of the symbol to which you want to attach the attribute and click once. Click again to return to normal editing mode.**

Tip: *Note that the attachment point—the item to which the attribute is attached—appears as red spot in the symbol.*

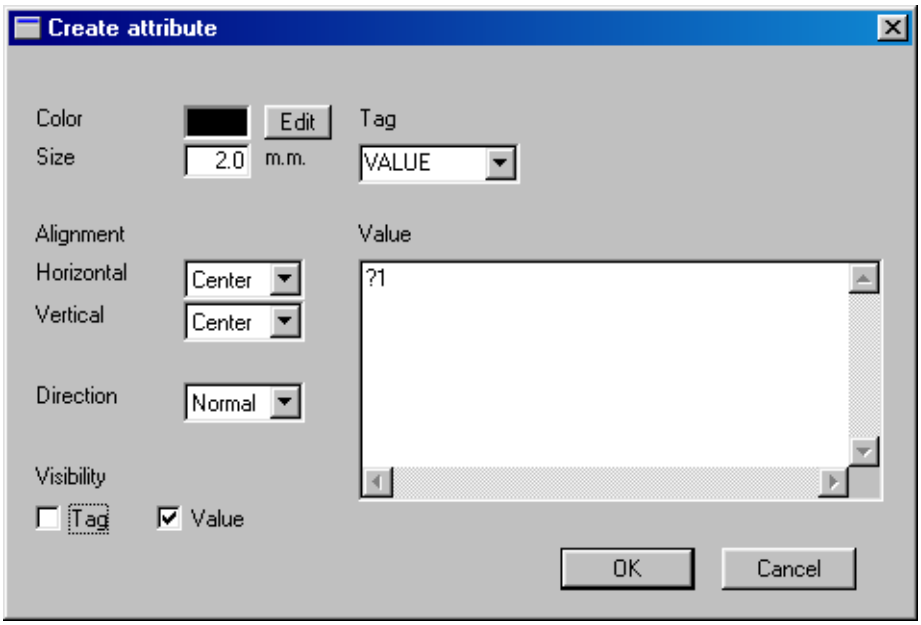
Our Stepper symbol should now appear as at right.



Our Stepper symbol still lacks an essential attribute: the value of the amount to increment the data input signal - the Step Value. For the moment, we have only added text to the symbol—this has no impact on the actual function of the symbol. To add provide a numerical step value to the macro, we need create another attribute:

- Click in the symbol editor toolbar, or select ***New→Attribute*** popup menu.

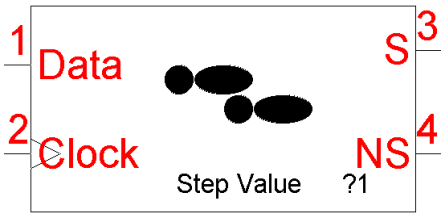
  The ***Create attribute*** dialog box opens.



- In the ***Tag*** drop-down list, select **Value**.

- In the ***Value*** field, type “?1”. This is the default value for the attribute.
- Under ***Visibility***, check the ***Value*** box only.
- Click OK when you have finished defining the attribute.
- The cursor becomes a ghost box indicating the size of the attribute. Position the cursor just beside the text “Step Value” and click once. Click again to return to normal editing mode.

Our Stepper symbol should now appear as at right.

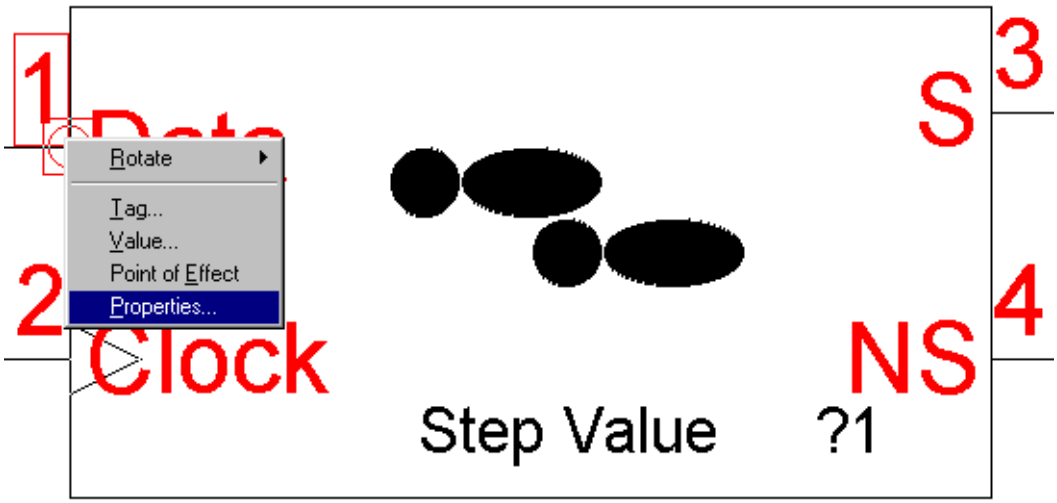


11.4.5 Modifying Existing Attributes

You can change the properties of an attribute you have already created while still in the symbol editor:

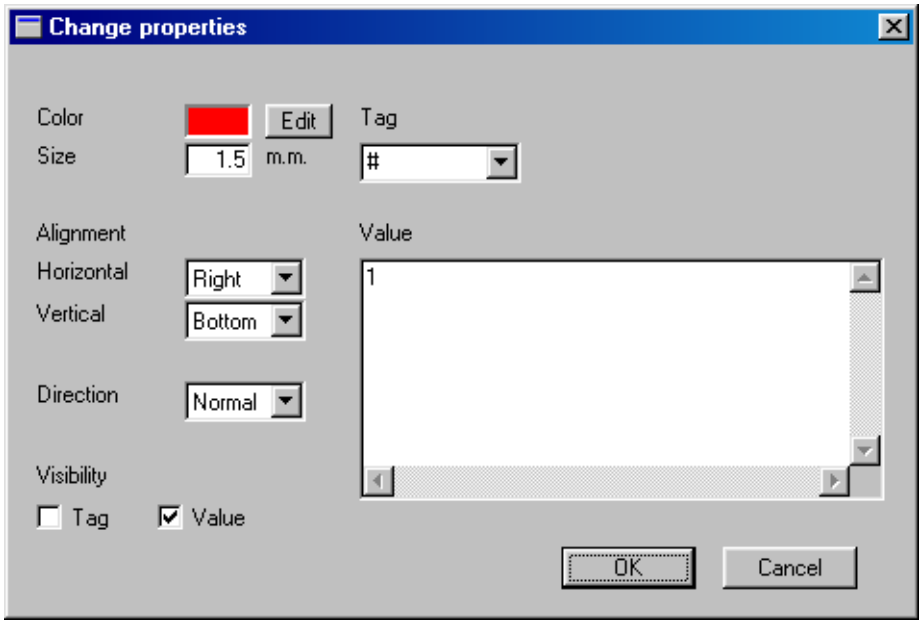
1 Select the attribute by clicking on it.

For example, in our Stepper example, we would like to remove the pin numbers. Select the pin number “1” by clicking on it.



2 Right-click the mouse. A popup menu appears, as shown above. Select *Properties*.

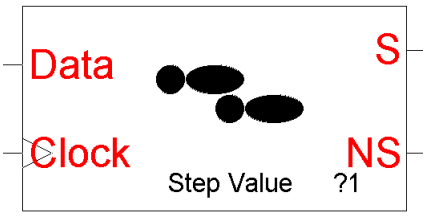
A Change properties dialog box appears.



Under ***Visibility***, uncheck the ***Value*** box. Click OK.

The pin number will disappear from the symbol.

In a similar manner, make all of the pin numbers disappear from the symbol. Our stepper symbol will now appear as at right.



Tip:

You can make attributes invisible, as we just did for the pin numbers, but they are still present in the symbol. To view invisible attributes, select ***View→Invisible attributes*** *from the main menu or from the popup menu.*

Note:

You may also change existing symbol attributes directly from the ST-Realizer worksheet where your scheme has been loaded. See “Changing a Symbol’s Attributes” on page 75.

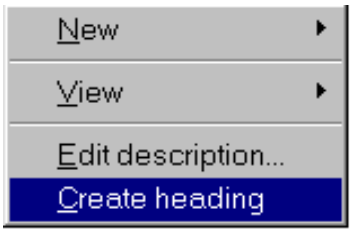
Recall from page 171 that there are four steps to complete the creation of a new user-defined symbol. We have now completed the first two steps in the creation of our Stepper symbol:

- The definition of the graphic aspect of the symbol, including assigning the required number of input and output pins, naming these pins and assigning pin data types and attributes.
- Editing the new symbol to further refine its appearance.

The next step is to create a macro header for the symbol, and then write, in assembler, the macros that will define how the symbol functions.

11.4.6 Creating the Macro Header

In the symbol editor popup menu, select **Create heading**.



The macro header for the new symbol is copied to the clipboard.

For our example symbol, Stepper, we generated two macro headers—iStepper and Stepper. iStepper looks like:

```
;*************************************************************************************************
;
;       istepperwbbww macro Data,tData,Clock,nClock,tClock,pClock,pnClock,ptClock,S,tS,NS,tNS,c5
;
;       Description: Initialize Word STEPPER
;*************************************************************************************************
```

In the iStepper macro, the “wbbww” after the “istepper” is a listing of the inputs and outputs—word (the Data input pin), clock input bit (the Clock input pin) and the two output pins (S and NS) are both word types because they were programmed to take the maximum valued input pin type (i.e. the word type). (See Naming Macro Instructions below for more information on the abbreviations used for data types.)

Once the macro header has been copied to the clipboard, create a new ASCII text file (using an ASCII text editor) and paste the macro header into the text file.

11.4.7 Creating the New User-Defined Symbol Macro

Once the macro header has been pasted into an ASCII text file, the next step is to write the rest of the macro for the new symbol. Writing macros requires a knowledge of the assembler programming language, and it is now within the scope of this document to instruct the user how to write assembler code. We will simply give some general guidelines to follow, and show how the assembler macro for the our example symbol, Stepper, was created.

General Rules

When you create your own macros for inclusion in ST-Realizer applications, you must follow the rules described below.

- Symbols using multi-type pins require two macros: one for the BIT type and one for the WORD type. Symbols only using BIT type pins require one macro call.
- The macro instructions are written in assembly language using any plain ASCII text editor. Remember to save the ASCII file as a **`.mac`** file (for ST6) or as an **`.inc`** file (for ST7). For

example, we could save our Stepper symbol as **`stepper.inc`**.

Naming Macro Instructions

The names of macro instructions are a concatenation of the following:

- The attribute value.
- The type of the input/output variable ranked by the pin number.

  The type of the input output variable uses the following convention:

  w for a WORD type

  b for a BIT type

  bb for a BIT clock input

  ww for a WORD clock input

  For example, a symbol that has two input pins, one which is BIT type and the other WORD type, and one output pin of WORD type and with the attribute CODE=Stepper, has the macro name: ***Stepperbww***.

Parameters

The macro name is followed by the macro parameters. A parameter set is used for each variable according to the signal type and function.

BIT type input and output pins have the parameters: variable name, bit number, variable type, where bit number is the position of the bit within a byte.

WORD type input and output pins have the parameters: variable name, variable type.

BIT type clock input pins have three parameters for the current clock variable: variable, bit number, variable type and three parameters for the previous clock variable: variable, bit number, variable type.

WORD type clock input pins have two parameters for the current clock variable: variable name, variable type and two parameters for the previous clock variable: variable name, variable type, where current and previous refer to the loop.

The parameter sets are ranked by the pin number.

11.4.8 Writing the Assembly Macro

Macro headers are created as requested on page 175. For our Stepper symbol, we requested an initialization macro header, called iStepper, and a main code macro header, simply called Stepper. These macro headers enumerate the parameter list for each portion of the code:

- for the initialization portion of the code, the parameter list is: Data, tData, Clock, nClock, tClock, pClock, pnClock, ptClock, S, tS, NS, tNS and c5.

- for the main portion of the code, the parameter list is: Data, tData,Clock, nClock, tClock, pClock, pnClock, ptClock, S, tS, NS, tNS and c5.

The final macro calls (i.e the contents of the ASCII file **stepper.inc**) will look like this:

```
;*******************************************************************************
;
;     istepperwbbww macro Data,tData,Clock,nClock,tClock,pClock,pnClock,ptClock,S,tS,NS,tNS,c5
;
;     Description: Initialize Word STEPPER
;*******************************************************************************

istepwbbww macro Data,tData,Clock,nClock,tClock,pClock,pnClock,ptClock,S,tS,NS,tNS,c5
  #if {tNS}
    ld A,#0FFH
    #if {{tNS eq TUBYTE} or {tNS eq TSBYTE}}
      ld NS,A
    #endif
    #if {{tNS eq TUINT} or {tNS eq TSINT}}
      ld NS,A
      ld {NS+1},A
    #endif
    #if {tNS eq TLONG}
      ld NS,A
      ld {NS+1},A
      ld {NS+2},A
      ld {NS+3},A
    #endif
  #endif
  mend

;*******************************************************************************
;
;     stepperwbbww macro Data,tData,Clock,nClock,tClock,pClock,pnClock,ptClock,S,tS,NS,tNS,c5
;
;
;     Description: WORD stepper
;*******************************************************************************
stepwbbww macro Data,tData,Clock,nClock,tClock,pClock,pnClock,ptClock,S,tS,NS,tNS,c5

  local labnr

  ;test rising edge on the Clock input
  bittjf Clock,nClock,E&labnr
  bittjt pClock,pnClock,E&labnr

  ;addition of 2 bytes
  #if {tS}
    #if {tData eq TCONST}
      ld A,#{low {Data}}
    #endif
    #if {{tData eq TUBYTE} or {tData eq TSBYTE}}
      ld A,Data
    #endif
    #if {{tData eq TUINT} or {tData eq TSINT}}
      ld A,{Data+1}
    #endif
    #if {tData eq TLONG}
      ld A,{Data+3}
    #endif

    ;#if {tin2 eq TCONST}
      add A,#{low {c5}}
    ;#endif
  #endif

  #if {{tS eq TUBYTE} or {tS eq TSBYTE}}
    ld S,A
  #endif
  #if {{tS eq TUINT} or {tS eq TSINT}}
```

```
    ld {S+1},A
  #endif
  #if {tS eq TLONG}
    ld {S+3},A
  #endif

  #if {{tS eq TUINT} or {tS eq TSINT} or {tS eq TLONG}}
    #if {tData eq TCONST}
      ld A,#{low {{Data} shr 8}}
    #endif
    #if {tData eq TUBYTE}
      clr A
    #endif
    #if {tData eq TSBYTE}
      clr A
      push CC
      bittjf Data,7,A&labnr
      cpl A
A&labnr:
      pop CC
    #endif
    #if {{tData eq TUINT} or {tData eq TSINT}}
      ld A,Data
    #endif
    #if {tData eq TLONG}
      ld A,{Data+2}
    #endif

    ;#if {tin2 eq TCONST}
      adc A,#{low {{c5} shr 8}}
    ;#endif
  #endif

  #if {{tS eq TUINT} or {tS eq TSINT}}
    ld S,A
  #endif
  #if {tS eq TLONG}
    ld {S+2},A
  #endif

  #if {tS eq TLONG}
    #if {tData eq TCONST}
      ld A,#{low {{Data} shr 16}}
    #endif
    #if {{tData eq TUBYTE} or {tData eq TUINT}}
      clr A
    #endif
    #if {{tData eq TSBYTE} or {tData eq TSINT}}
      clr A
      push CC
      bittjf Data,7,D&labnr
      cpl A
D&labnr:
      pop CC
    #endif
    #if {tData eq TLONG}
      ld A,{Data+1}
    #endif

    ;#if {tin2 eq TCONST}
      adc A,#{low {{c5} shr 16}}
    ;#endif

    ld {S+1},A

    #if {tData eq TCONST}
      ld A,#{low {{Data} shr 24}}
    #endif
    #if {{tData eq TUBYTE} or {tData eq TUINT}}
      clr A
```

```
    #endif
    #if {{tData eq TSBYTE} or {tData eq TSINT}}
      clr A
      push CC
      bittjf Data,7,G&labnr
      cpl A
G&labnr:
      pop CC
    #endif
    #if {tData eq TLONG}
      ld A,Data
    #endif

    ;#if {tin2 eq TCONST}
      adc A,#{low {{c5} shr 24}}
    ;#endif

    ld S,A
  #endif
  #if {tNS}
    copyww S,tS,NS,tNS
    comw NS,tNS
  #endif


E&labnr:
  mend
```

11.4.9 Adding New User-Defined Symbols to a Library

You can save your new user-defined symbol to a library by following these steps:

1 **In any ASCII text editor, open the library file to which you wish to save your new symbol.**
For example, if you wish to save your symbol to the main library, open `<root folder>\Lib\st72lib.mac` (the ST6 main symbol library) or `<root folder>\Lib\st72lib.inc` (the ST7 main symbol library).
If you wish to save the symbol to a new library, create a new ASCII file in the `<root folder>\Lib` folder and call it an appropriate name, such as `my_symbols.mac` (for ST6) or `my_symbols.inc` (for ST7).

2 **Copy and paste the new symbol's ASCII macro (i.e. the contents of the ASCII macro file you created on page 187) either to your newly created library file, or to the end of the main library file above.**

3 **Save the library file.**

Note: It is always wise to save your new symbol macros and any modified or created libraries to a back-up folder, or on a floppy disk. In the event that you need to reinstall ST-Realizer, the root folder will be overwritten, and all of your customized symbol libraries will be replaced by the ST-Realizer default libraries.

4 **Next, save the symbol graphic by copying and pasting the new symbol graphic from within the scheme where you created it to a new, blank scheme. From this new**

scheme containing the new symbol, select *File→Save as.. `filename`*`.lib` where *`filename`* is the name of the symbol (for example, *stepper*).

Tip: *When you save a scheme as a library, you will copy all of the symbols in that scheme to the library. Therefore, to save only the new user-defined symbol to the library, it needs to be in a scheme by itself.*

Each time you wish to use the symbols that you have created, from within the project where you are working, perform the following:

1 **Select Project→Hardware settings.**

2 **In the *Hardware settings* dialog box, in the *Include file* field, ensure that the library that you saved your symbol to appears in the list of files:**

- `<root folder>\Lib\st72lib.mac` (for ST6) or `<root folder>\Lib\st72lib.inc` (for ST7) if you saved your symbol to a main library.
- `<root folder>Lib\`*`filename`*`.mac` (for ST6) or `<root folder>Lib\`*`filename`*`.inc` (for ST7) where *`filename`* is the name of the new symbol library you created above.

12 CUSTOMIZING ST-REALIZER

A number of ST-Realizer features can be customized to suit your way of working. These include:

- **Automatic work saving**
- **Screen preferences**
- **Worksheet layout preferences**
- **Printing options**
- **Symbol layout preferences**
- **Wire drawing options**
- **Attribute display preferences**
- **Toolbar contents**

To customize these features, proceed as follows:

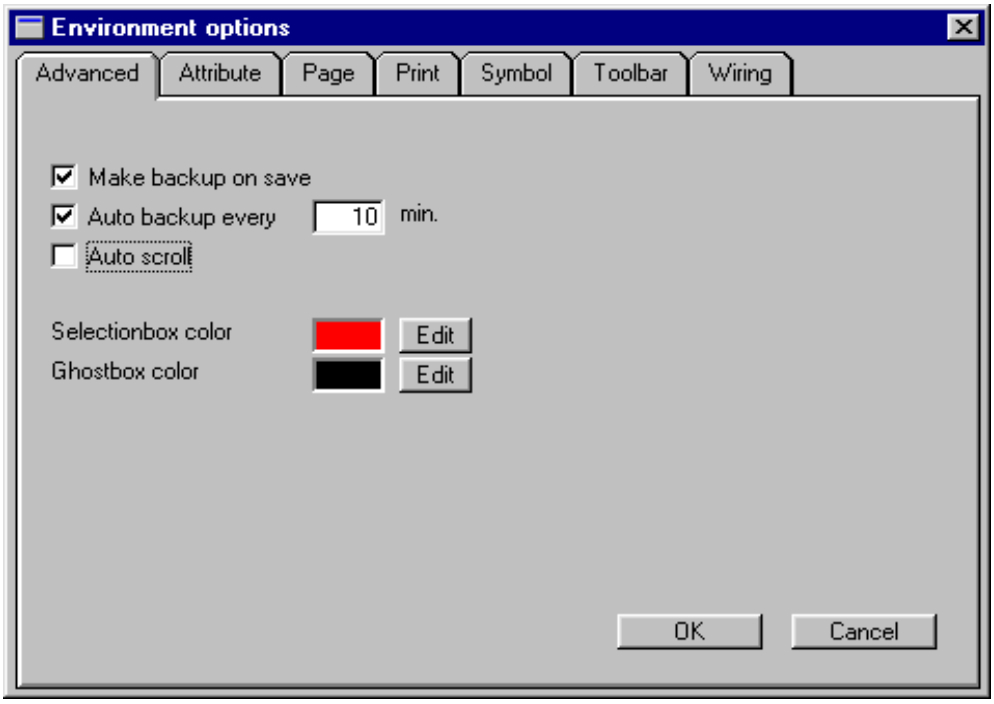
1 Select *Options→Environment* from the main menu bar.

The **Environment Options** dialog box opens showing a number of tabs that direct you to the following daughter dialog boxes:

- Advanced
- Attribute
- Page
- Print
- Symbol
- Toolbar
- Wiring

12.1 Automatically Saving Your Work and Setting Screen Preference.

ST-Realizer can back up and save your work automatically. This option can be chosen in the daughter dialog box accessed by clicking the **Advanced** tab:



1 Select the backup options that you wish.

To create a backup file every time you save your work:

- Check the ***Make backup on save*** check box by clicking it.

This creates a copy of the file before your latest changes are saved. If a backup file already exists, the new backup file overwrites it.

To create a backup file at regular intervals:

- Check the ***Auto backup every*** check box by clicking it, and enter an interval (in minutes) in the text box. If you want to change this, overtype the displayed value.

This causes a copy of the file to be saved periodically.

2 Set/Change the screen preferences.

To change the ***Ghost box*** or ***Selection box*** colors:

- Click the appropriate ***Edit*** button and select the new color from the displayed palette.

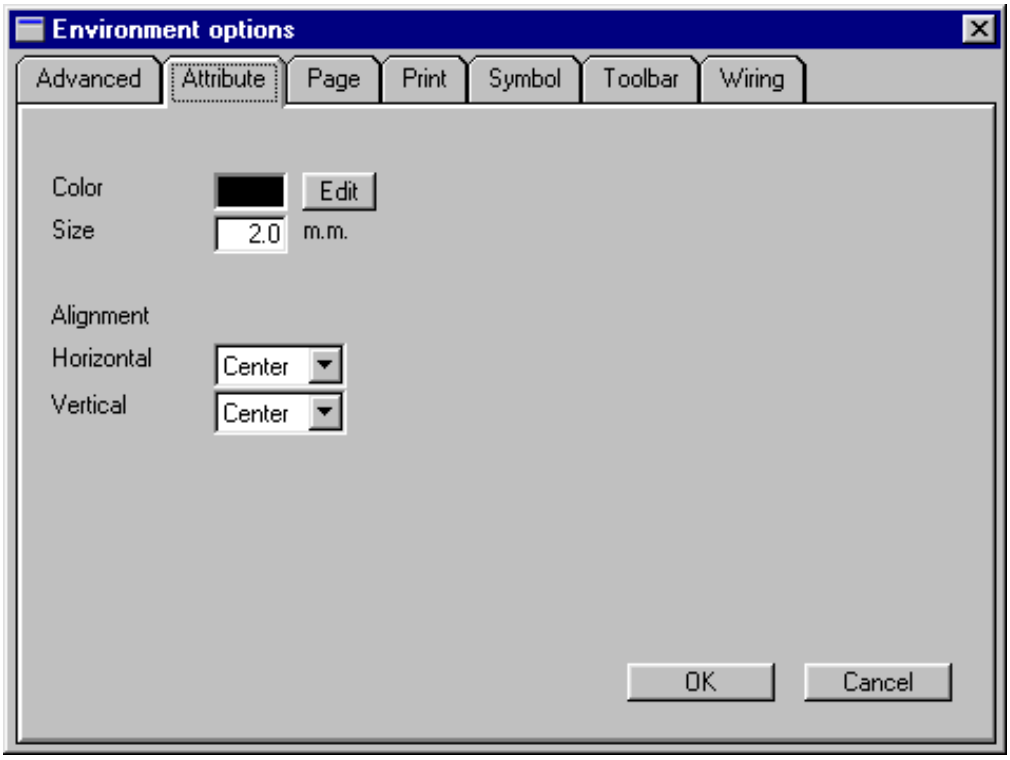
3 Click Ok when you have finished.

12.2 Attribute Display Preferences

In this tab, you can change the default settings for the display of attributes on the worksheet.

1 In the Environment Options dialog box, click the *Attribute* tab.

The **Attribute** dialog box opens.



2 To change the attribute label color, click the *Edit* button and select the new color from the displayed palette.

3 To change the attribute label font *Size*, overtype the appropriate value in the text box.

4 To change the attribute *Alignment*, select a new value from the appropriate drop-down list.

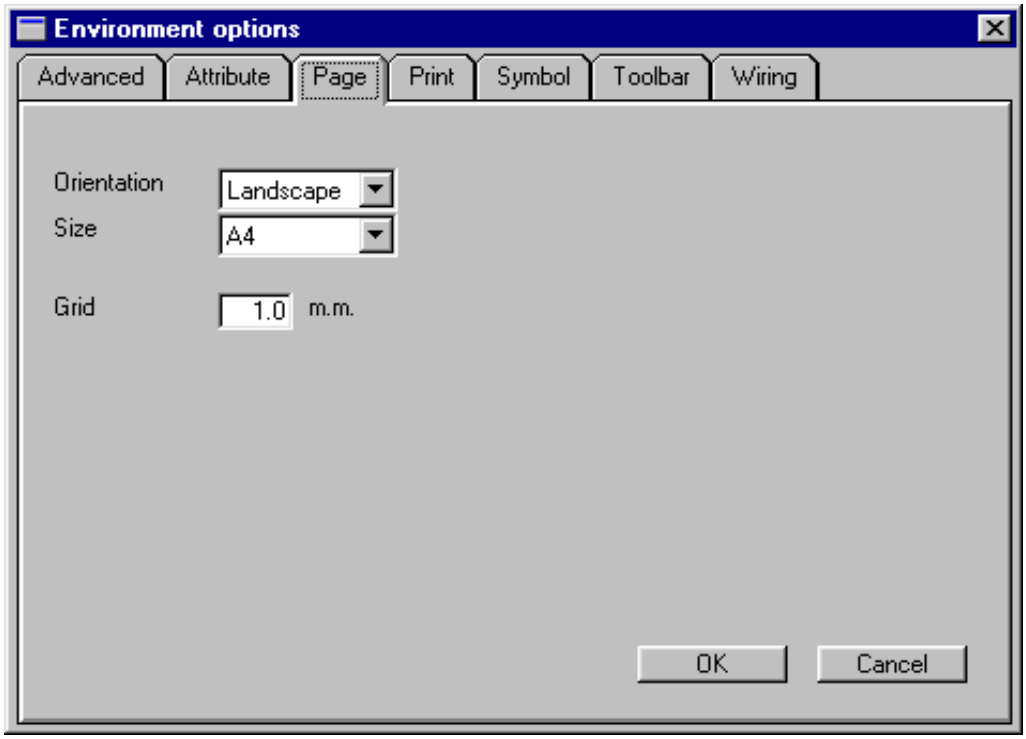
5 Click OK when you have finished.

12.3 Worksheet Layout Preferences

In this dialog box, you can change the size and orientation of your scheme worksheet. These characteristics are visible on screen and when you print a scheme.

1 In the Environment Options dialog box, click the *Page* tab.

The **Page** dialog box opens.



2 To change the worksheet orientation, select the appropriate option in the *Orientation* drop-down list.

3 To change the worksheet size, select a new size from the *Size* drop-down list.

4 To change the grid distance overtype the appropriate value in the *Grid* box.

5 Click OK when you have finished.

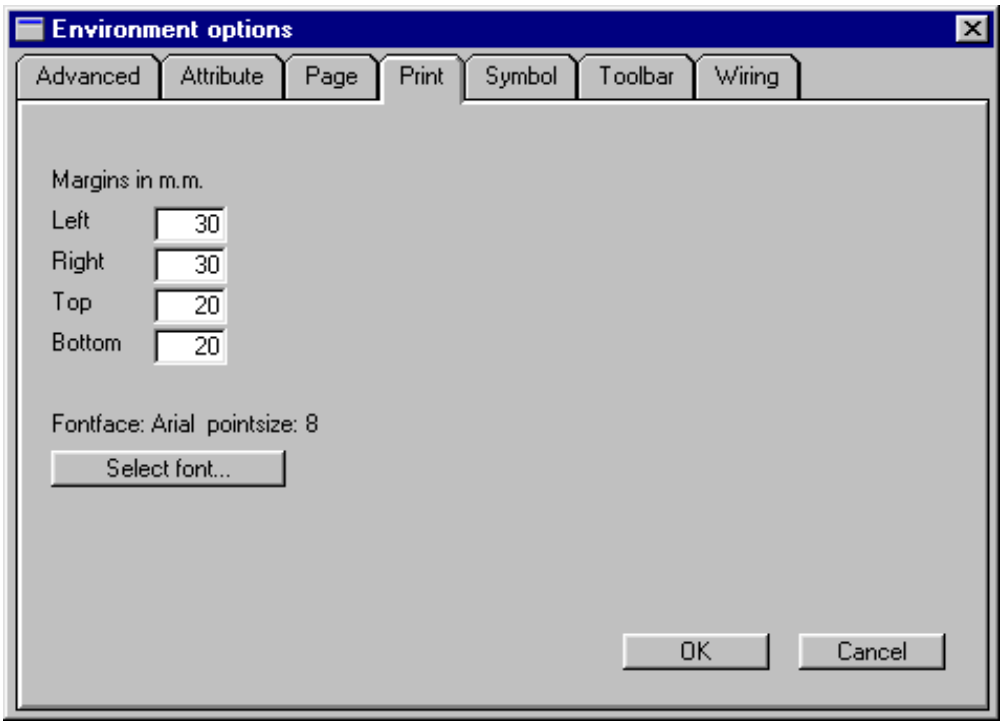
12.4 Printing Options

In this dialog box, you can change the margins and the font of your printed scheme page.

These characteristics are visible on screen and when you print a scheme.

1 **In the Environment Options dialog box, click the *Print* tab.**

The **Print** dialog box opens.



2 **To change the margins, overtype the values in the appropriate boxes.**

3 **To change the font, click the *Select font...* button, and select the new font from the list.**

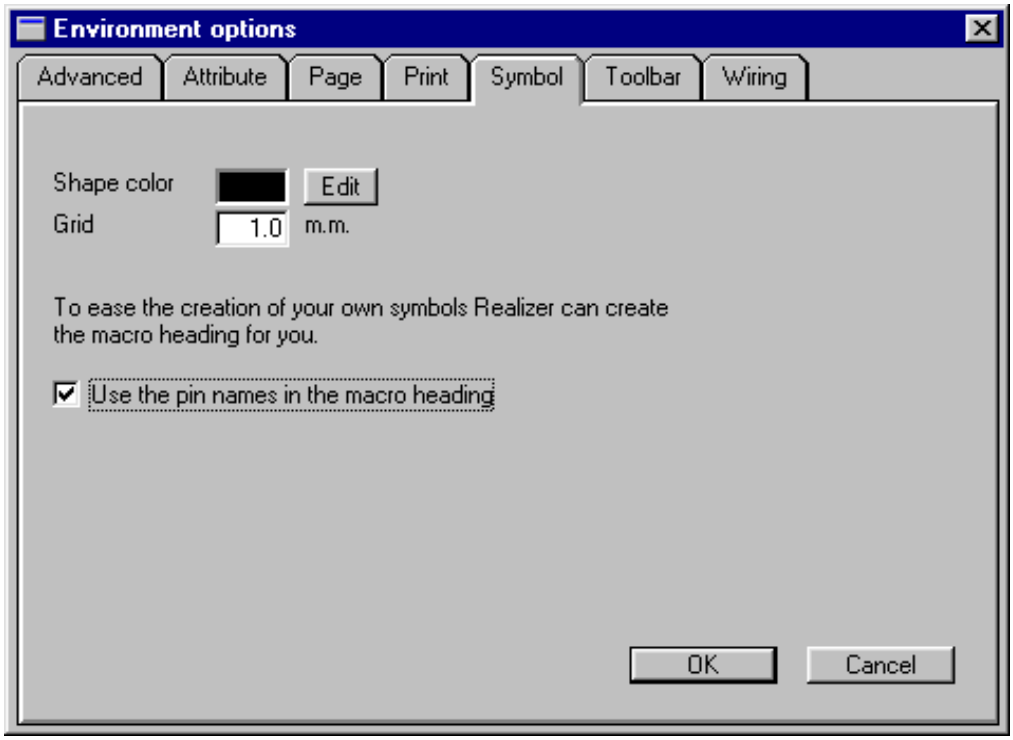
4 **Click OK when you have finished.**

12.5 Symbol Layout Preferences

In this dialog box, you can change the way symbols are displayed on the worksheet.

1 In the Environment Options dialog box, click the *Symbol* tab.

The **Symbol** dialog box opens.



2 To change the color of symbols, click the *Edit* button and select the new color from the displayed palette.

3 To change the grid distance within symbols, overtype the appropriate value in the *Grid* box.

Each symbol you create is normally linked to one or more macros. These macros have headings with a specific format. For details, see “Naming Macro Instructions” on page 188.

4 To allow pin names to be used in macro headings click the corresponding check box.

5 Click OK when you have finished.

12.6 Customizing Toolbars

Toolbar buttons provide you with quick access to frequently-used commands. Most of the ST-Realizer commands have their own, predefined buttons. You can change the ST-Realizer toolbars by:

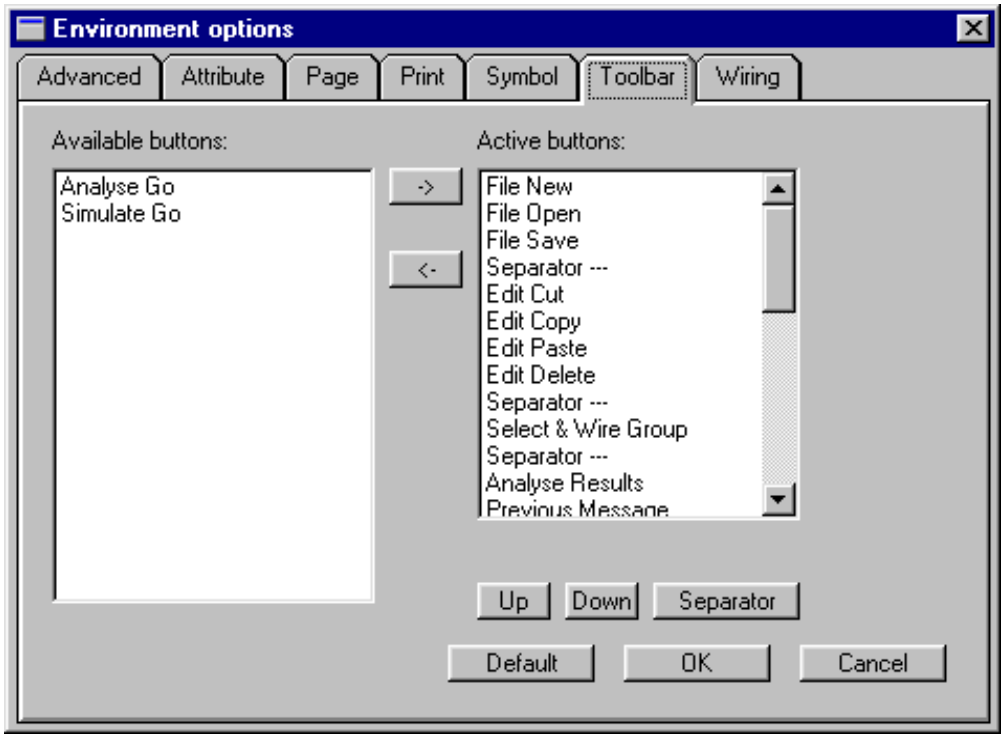
- **Adding or deleting toolbar buttons.**
- **Placing separators between toolbar buttons.**
- **Changing the order of toolbar buttons.**
- **Restoring the default toolbar.**

Note: Note that the changes you make to the toolbar are not implemented until you open a scheme.

The following paragraphs describe how to perform these tasks.

1 In the Environment Options dialog box, click the *Toolbar* tab.

The **Toolbar** dialog box opens. The ***Available buttons*** box lists the toolbar buttons that are available but are not currently being used. The ***Active buttons*** box lists the toolbar buttons that are currently being used.



12.6.1 Adding and Deleting Toolbar Buttons

1 To add a button to the toolbar:

- In the ***Active buttons*** box, select the button to the left of which you wish to add the new button.
- In the ***Available buttons*** box, click the name of the button you want to add, then click .

2 To delete a button from the toolbar, click its name in the Active buttons box, then click .

3 Click OK.

12.6.2 Placing Separators Between Toolbar Buttons

You can place separators between toolbar buttons, so that you can group the buttons as you like. For example, you could group the edit functions: Cut, Copy and Paste buttons. Placing a separator widens the space between two buttons.

To place a separator:

1 In the Active buttons box, click the button to the left of which you want to place a separator.

2 Click the *Separator* button, then click OK.

12.6.3 Changing the Order of Toolbar Buttons

1 In the *Active* buttons box, click the button you want to move

2 Click *Up* to move the button one place to the left, or *Down* to move the button one place to the right.

3 Repeat step 3 until the button is in the required position.

4 Click OK when you have finished ordering the buttons.

12.6.4 Restoring the Default Toolbar

1 Click Default.

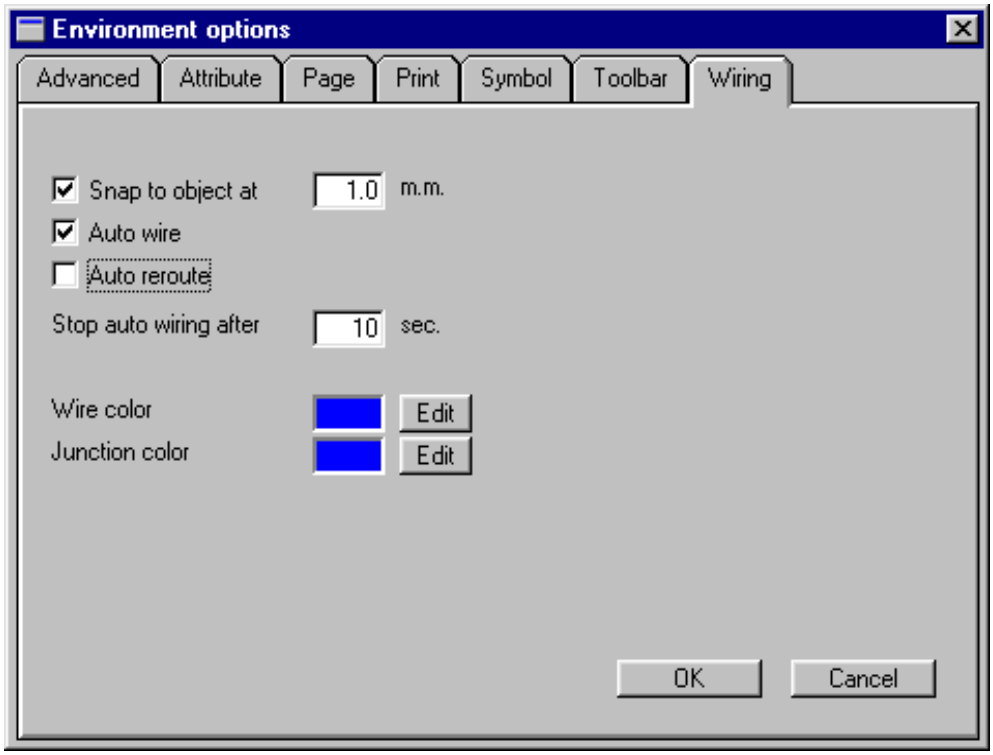
2 Click OK.

12.7 Wire Drawing Options

You can change wire characteristics and snapping distances.

1 In the Environment Options dialog box, click the *Wiring* tab.

The **Wiring** dialog box opens.



2 To change the snapping distance, click the corresponding check box and overtype the appropriate value in the text box.

3 To enable the auto wiring and/or auto reroute features, click the appropriate check box.

- With ***Auto wiring***, ST-Realizer draws the wires by automatically choosing the shortest path between the symbols to be connected, creating corners where required.
- With ***Auto reroute***, ST-Realizer automatically reroutes wires when a symbol is moved.

4 To change the color of wires or junctions, click the appropriate *Edit* button and select the new color from the displayed palette.

5 Click OK when you have finished.

APPENDIX A: VARIABLES AND ATTRIBUTES

This appendix provides you with quick access to the type of information you'll need when you create ST-Realizer applications. This information includes:

- A list of the type of variables you can define, and the rules that apply to variables.
- A list of the attributes you can place on symbols and wires.

A1 Variable Types and Rules

ST-Realizer lets you define the following data types:

Table A1 Data Types

Name	Range	Data type	Number of bytes
BIT	0,1	Boolean Bit	
UBYTE	0...255	Unsigned byte	1
SBYTE	-128...+127	Signed byte	1
UINT	0...65535	Unsigned integer	2
SINT	-32768...32767	Signed integer	2
LONG	-2147483648.. .2147483647	Signed long	4
WORD	Represents any type, except BIT	Any, except Boolean	1 through 4

Most of the symbols included with ST-Realizer support multiple-type pins. This means that any variable types can be assigned to these pins. For example, the AND2 symbol can be used as an AND or either two BIT or two UINT variables.

ST-Realizer handles multiple type variables in two groups:

- The BIT variables.
- The UBYTE .. LONG (=WORD) variables.

You cannot mix these variable groups, for example, an AND2 symbol cannot be used to perform an AND on a BIT and a UBYTE variable.

Note that the type WORD covers any data type other than BIT.

To define a variable type, you attach an attribute with the ***Tag*** TYPE and the name of the variable type in the ***Value*** field.

Note: A LONG variable uses 4 bytes in RAM, while a UINT or SINT variable uses 2 bytes and an UBYTE uses 1 byte. To save RAM space, it is therefore recommended that you use the LONG variable range only when needed.

A1.1 Type Inheritance

Multiple-type pins support type inheritance: this means that the output pin type is the same as the greatest of the input pin types.

The following picture shows how type is inherited by the AND2 symbol.

Note that, if the expected result is outside the expected range, you must set the appropriate type for the output pin by placing a TYPE= attribute on that pin. For example, if a multiplication is performed between the values 124 and 245, which are both UBYTE type variables, the final result will be 30380, which is a SINT type variable. Thus the SINT type must be attributed to the output wire.

Note: Operations between BIT and WORD variables are not allowed.

A1.2 Type Overruling

You can overrule type inheritance rules. By placing an attribute with the ***Tag*** TYPE on the output wire, you can set the output of a symbol to a specific variable type.

This feature can be useful when you use the MUL symbol to multiply two values. When you multiply two SINT variables an overflow may occur: 2000 * 1500 becomes -14656 instead of the expected 3000000. By putting a TYPE=LONG attribute on the output wire of the MUL symbol, this overflow is prevented (the value 3000000 can easily fit into a LONG variable). For example:

Another example of type overruling is where the DIV symbol (divide) is used to divide two SINT variables. This results in a value that fits onto an SBYTE variable. By using the TYPE= attribute a smaller variable type can be defined, generating more efficient and faster code:

A2 Attribute Types

The following tables list the types of attribute that you can place on pins and symbols, and describe the available values and meanings.

A2.1 Pin Attributes

Tag	Description
#	Pin numbering: # = 0..x Specifies the parameter sequence.
TYPE	Specifies the variable type of a pin. The following table lists the available values for TYPE tags and their ranges: **Value** **Range** EVENT 0,1 BIT 0,1 UBYTE 0 .. 255 SBYTE -128 .. 127 UINT 0 .. 65535 SINT -32768 .. 32767 LONG -2147483648 .. 2147483647 WORD = any type except BIT When one of these values is specified, the connecting wire must be of the same type. For input pins the attribute: TYPE = WORD is available. This allows type UBYTE .. LONG to be connected to the pin. For output pins a combination of input pins can be used to define the output type: MAX a,b,c,...: a,b,c,.. are pin numbers. MIN a,b,c,... The MAX value returns the largest of the input pin types and the MIN value return the smallest. For example, MAX1 will return the type of input pin 1. MAX12, will return the largest of the input pin types of pins 1 and 2. Non volatile allocation: ALLOCATION = "FIXED" . Allocates a non volatile variable with the same variable type as the pin. Local variables: ALLOCATE = x 'x' extra variables are added, the variable type of the pin is used.

A2.2 Symbol Attributes

Tag	Description
ALLOCATEBLOCK IN	Defines the memory space (RAM or EEPROM).
ALLOCATION	By adding the ALLOCATION = FIXED attribute to an output pin, ST-analyser will allocate a non-volatile variable (usually a EEPROM variable) to it. This variable is added to the macro parameter list after the list of parameters of that particular pin. This enables you to make your own non-volatile symbol.
BLOCK SIZE IN UNITS	Defines the size of the block in units (variables).
CODE	Defines a macro that is executed inside the main execution loop. Enter the macro name in the **Value** field.
COMMENT	The value of this attribute is used in the report file.
COUNT PIN	See “DEVICE”
DEVICE	DEVICE attributes apply to macros associated with some particular symbols. They are automatically specified when you select the symbol. They extend the macro definition with additional information added in the macro argument list after the pin and value attribute parameters. The available DEVICE values are: DEVICE = INPUT, which extends the macro argument list with the input port parameters. In this case the attribute NAME must be used to associate a hardware port. Examples: **input**, **inputlatch** symbols. DEVICE = INPUTREG, is used to enable a hardware connection to an Input register. DEVICE = INPUTALL, is used to enable a hardware connection to any hardware resource. DEVICE = OUTPUT, which extends the macro argument list with the output port parameters. In this case the attribute NAME must be used to associate a hardware port. Examples: **output**, **outputlatch** symbols. DEVICE = OUTPUTREG, is used to enable a hardware connection to an output register. DEVICE = OUTPUTALL, is used to enable a hardware connection to any hardware resource..

Tag	Description
DEVICE (cont'd)	DEVICE = TIMER, which extends the macro argument list with a time value, a time variable and a system tick variable. The time value is optional (attribute "TIME = 2:00.00"). The type of the time variable is determined by the constant time value or the attribute "TIMEPIN = x", where x is an input pin number. The TIMER attribute results in the following macro call extensions: If the TIME = attribute exists: time,Ttime,timer,Ttimer,tick,Ttick If the TIMEPIN = attribute exists: timer,Ttimer,tick,Ttick Examples: **timf**, **timv** symbols. DEVICE = MICROSEC, is used to create a micro second delay, using timebase with the attribute TIMEBASE = x . DEVICE = COUNTER, which extends the macro argument list with an additional variable and its type. This variable is used as an internal variable (the counter value). The type of the internal variable is defined by the "COUNTPIN = x" attribute, where x = a pin number. Examples: **countf**, **countv** symbols. DEVICE = SHIFT, which extends the macro argument list with an additional variable and its type. This variable is used as an internal variable (the shift value). The type of the internal variable is defined by the "SHIFTPIN = x" attribute, where x = a pin number. Example: **shift** symbol.
ICODE	Defines a macro that is executed once, before the main program loop is executed, to initialise the symbol's properties. Enter the macro name preceded by **i** in the **Value** field.
LABEL	Used to link two wires by means of a name or to name an object.
NAME	See "DEVICE"
OCODE	Defines a macro that is executed at the end of the execution loop. Enter the macro name preceded by **o** in the **Value** field.
SCHEME	Subscheme attribute; specifies the name of the file (*.sch) associated with the subscheme
SHIFT PIN	See "DEVICE"
TABLE	Extends the macro parameter list with a reference to a ROM table, the number of records in the ROM table and the default value of the table: table,nrOfrecs,defval The tag TABLETYPE can also be used with the value INDEX or LOOKUP, to generate an indexed table or a lookup table respectively. The "TABLE =" attribute defines the filename of the table.

Tag	Description
TIME	See “DEVICE”
TIME PIN	See “DEVICE”
TXT	Specifies ASCII data to be entered as plain text.
UNIT TYPE	Type of a variable, from BIT through WORD, and also MIN and MAX.
VALUExxxx	The values of attributes with the tag: VALUE, are added after the pin parameters. VALUEx = x , is used to create a constant.
FAMILY	FAMILY="ST623" , used to identify hardware specific symbols. This string is compared with the device string when there is a partial compare the symbol is excepted by the analyzer.

APPENDIX B: SAMPLE APPLICATIONS

B1 Coded Lock Application

Applications generally include more and more security features. These aim to insure data confidentiality, access control or identify users. Non-volatile memory is usually required to store the identification or secret code. With the embedded EEPROM provided on the ST6 or ST7 MCU, one-chip solutions can be developed with the associated cost and density advantages.

The following example shows a coded lock system that was developed using ST-Realizer.

B1.1 Application Overview

This application manages a coded lock for a door. A secret code is loaded into the lock system to allow the door to be unlocked only if that code is entered. When loaded, the secret code is stored in the non-volatile memory (EEPROM) embedded in the ST6 or ST7 MCU. This ensures that the data is retained, even after a voltage cut-off, and insures security.

The application includes the following features:

- Secret code recording (3 digits).
- Recognition of entered access codes.
- Door lock control.

All the described functions of the system are managed in the final application under software control by the MCU.

Only the core of the application, the secret code storage in EEPROM and the code recognition, are described as a generic base for various applications. Any type of user interface, such as keyboard, IR or RF could be used, while the output signal can activate any kind of circuit. For this example, it is assumed that the user interface provides the following 4 inputs:

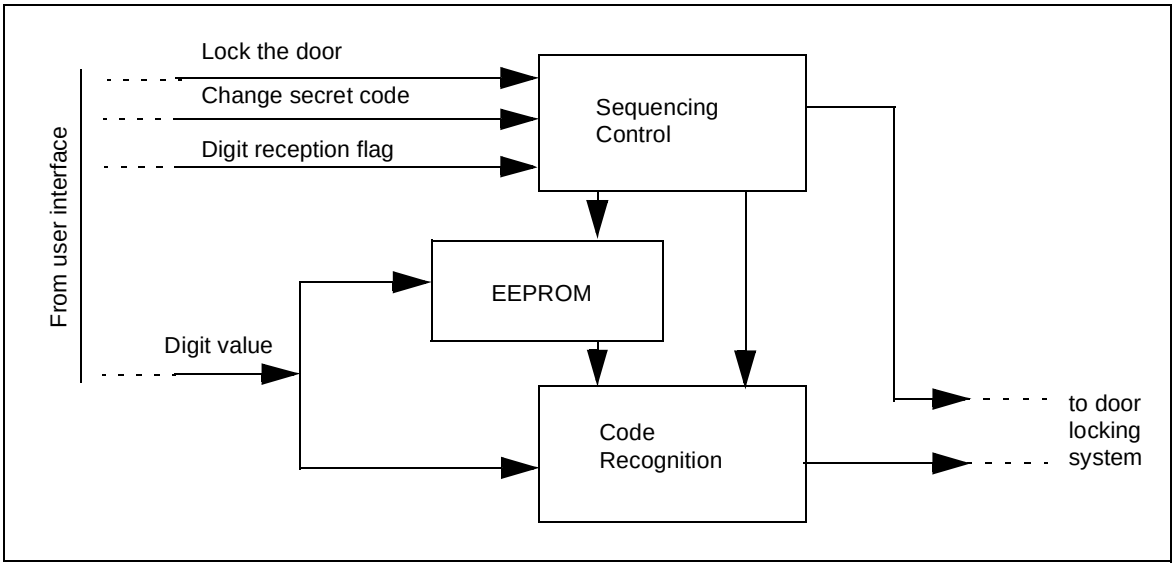
- Digit reception flag
- Digit value (0..9)
- Change secret code
- Lock the door

The digits used for the code are received serially, and are announced by the activation of a flag.

B1.2 Functional Description

Figure B1 below shows the application block diagram. Note that the application is restricted to its core: operation sequencing, the secret code storage in the EEPROM and the code recognition.

Figure B1 Coded Door Lock Block Diagram



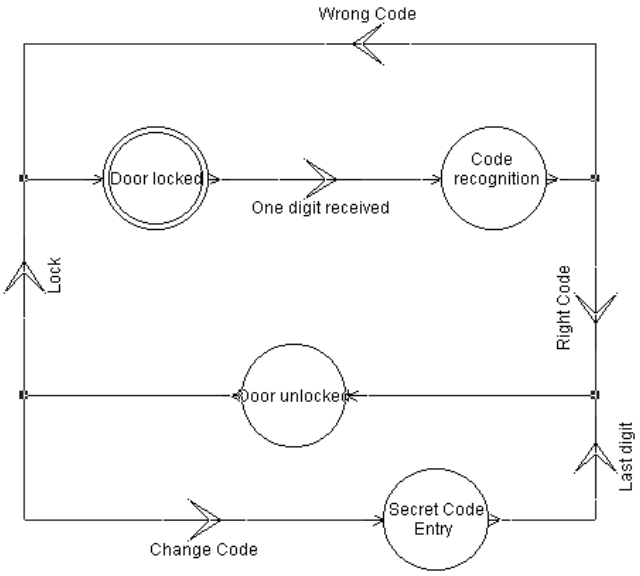
B1.3 Sequencing Control

The most important feature of this application is the sequencing control of all its operations. Two main items must be considered: the transition between the different working modes (Door locked, Access code entry, Door unlocked and Secret code entry) and the management of the serial flow of the numeric values received from the user interface.

The transition between the working modes is managed by a state machine, which manages four states:

- Door locked
- Code recognition
- Door unlocked
- Secret code entry

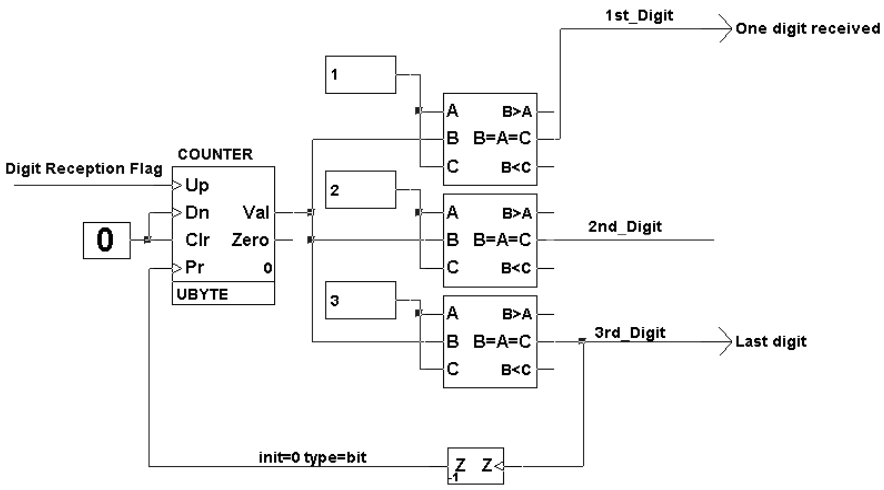
Figure B2 Sequencing Control State Machine



The conditional occurrences are generated through either the user interface (*Lock, Change secret code)*, or some functional sub-blocks of the application (*One digit received, Right code, Wrong code, Last digit*). For example, the occurrence of the condition *One digit received* in the Door locked state initiates the process code recognition.

The management of the serial flow of the digits entered (0..9) is carried out using a received digits counter (Figure B3). This stores the 1st, 2nd and 3rd digits received (in the case of a 3-digit code), in associated memory locations. When the 3rd digit is received, the counter is reset. In addition, the reception flags of the 1st digit and 3rd digit are used in the state-machine to initiate the code recognition and conclude the Secret code entry respectively.

Figure B3 Management of Digit Reception



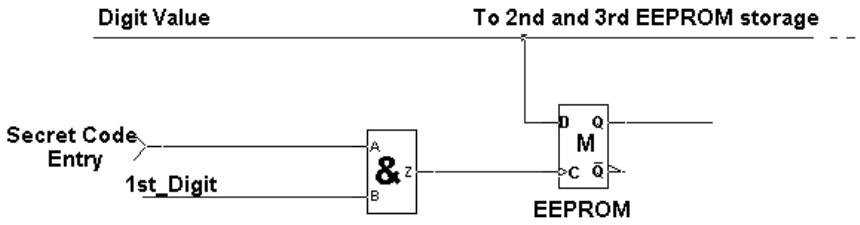
B1.4 Secret Code Storage in the EEPROM

ST-Realizer enables the EEPROM non-volatile memory to be managed as a standard bi-stable component: R/S Latch, D Latch, Shift Register or Counter.

Each EEPROM location is identified by a specific D Latch component providing the non-volatility feature. This component, provided within the ST-Realizer library, has the advantage of being multitype. This means the same symbol can be used whatever the input type: Bit, Byte, Word, Integer or Signed variable.

The data input comes directly from the interface as the digit value, in one byte. The number of D Latches must be equal to the number of digits used for the secret code, that is 3. There is a specific clock for each of the 3 D Latches, controlled by the sequencing control module. Data can be written (Clock activated) to a D Latch only if the active mode is Secret code entry, and if the received bit corresponds to this location (Figure B4). The clock is generated through an AND function between the Secret code entry mode and the ranking of the received digit. This ranking is issued by the counter used for serial flow reception.

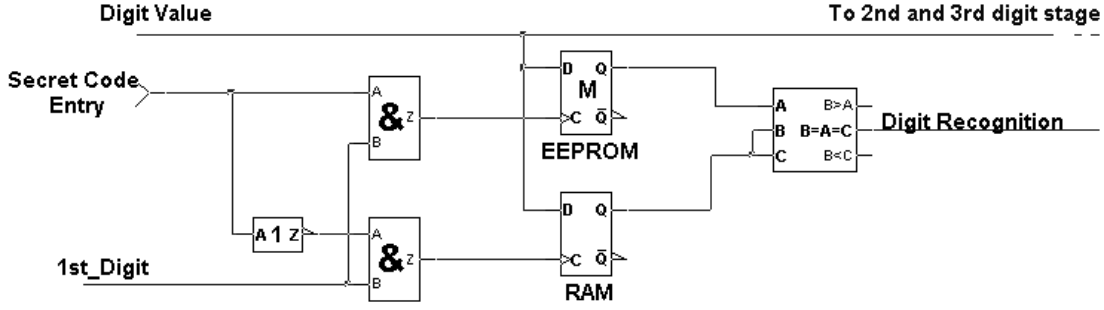
Figure B4 Digit Storage in EEPROM



B1.5 Access Code Entry and Recognition

The digits are received serially, as in the Secret code entry mode, and stored in Volatile Memory locations (RAM). The RAM locations are described with standard D Latch symbols. The input/output of this component are exactly the same as for the non-volatile components allowing a clear analogy between the functional description of the two modules. The data input channel is the same while the clocks are logically validated, except if the Secret code entry mode is activated (Figure B5).

Figure B5 Digit Management for Code Recognition



Code recognition is performed by a one-to-one comparison between the values stored in EEPROM and RAM.

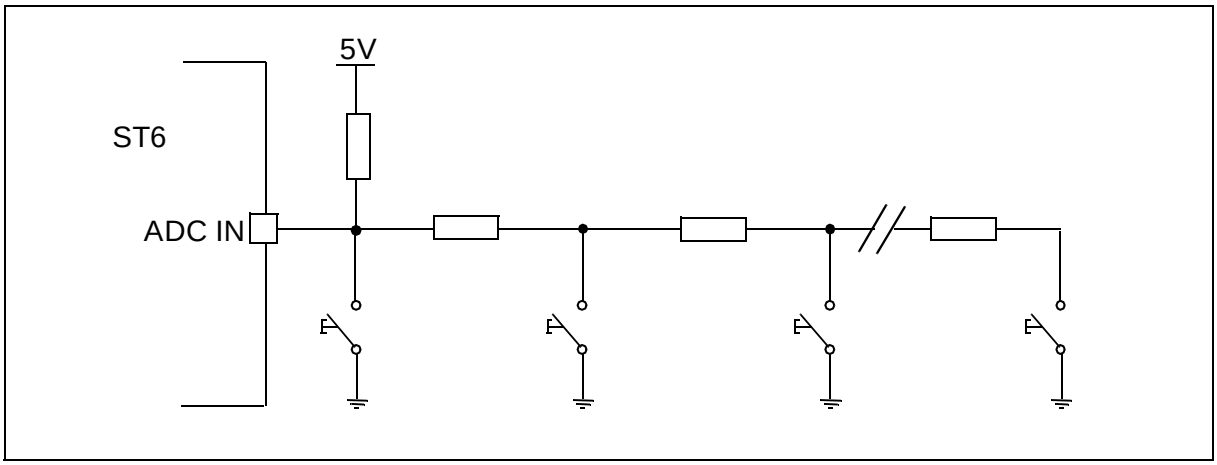
B2 Analog Multiple Key Decoder

One of the main uses of the ST on-chip Analog to Digital Converter (ADC) is to decode keys through one I/O port pin. The keys are connected to the converter input using a resistive voltage divider. This method is useful, since it only requires one I/O pin, whereas a traditional matrix keyboard requires a high number of I/O pins.

B2.1 Application Overview

The basic circuit of the decoder consists of a pull-up resistor connected to the ADC input, with the first key directly switching to ground. The following keys are then connected in sequence to the ADC input through serial resistors. The combination of the pull-up resistor, the serial resistors and the pressed key form a resistive voltage divider (Figure B6).

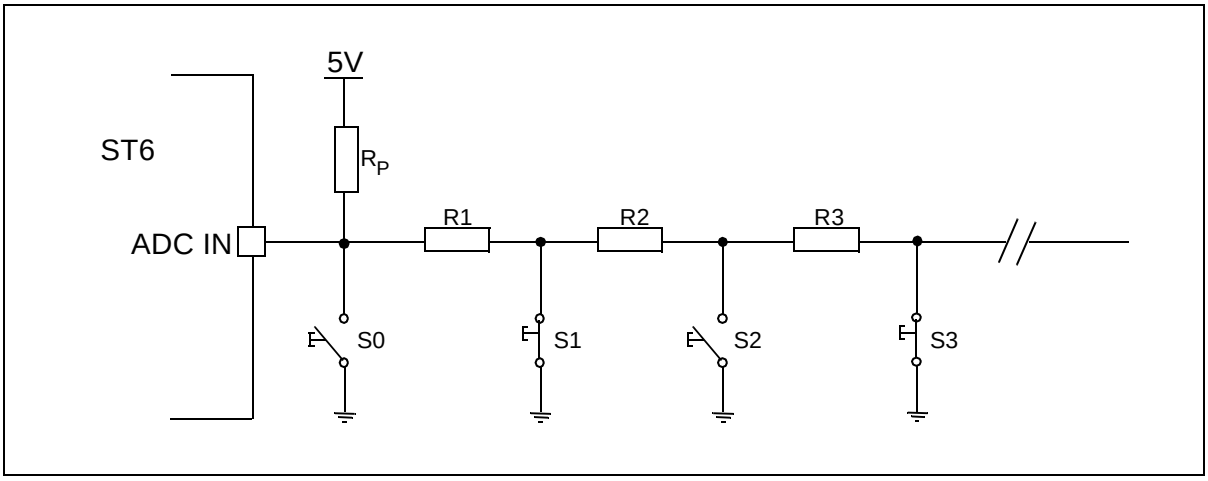
Figure B6 Analog Keyboard Resistor Key Matrix



When a key is pressed, the voltage at the ADC input is given by the activated voltage divider, generating a different voltage at the ADC input for each key that is pressed. If the top key is pressed, the voltage measured is zero while the default voltage at the ADC input (if no key is pressed) is Vdd.

This analog voltage is converted by the ADC and the digital output value is used to determine which switch is closed. If more than one key is pressed at the same time, the key detected is the key in the chain closest to the ADC input. This allows the keys in the keyboard to be prioritized (Figure B7).

Figure B7 Multiple Key Press



Depending on the identified key, a direct signal activation can be achieved or a selective jump in the program flow can be performed.

B2.2 The Keyboard

The serial resistors are selected in order to give an equal distribution of voltage between Vdd (No key pressed) and Vss (Last key pressed) for each switch combination, in order to give the best noise margin between keys. For n keys, the resistor values should be selected so that the voltage for the second key from top is Vdd/n, for the 3rd 2Vdd/n, for the 4th 3Vdd/n and for the nth (n-1)Vdd/n.

The maximum number of keys is limited by the precision of the resistors that provide a voltage value for each key pressed, within a margin of error either side of the theoretical value.

For a 10-key system, the values (in Ohms) given in Table B2 are used for the resistor network. Taking into account +/- 2% resistors, the voltage values and conversion results given in Table B3 can be obtained. Vmin is obtained when the serial resistors are at their minimum value and the pull-up resistor Rp is at its maximum value. Vmax is obtained when the serial resistors are at their maximum value while the pull-up resistor Rp is at its minimum value.

Table B2 Used Resistors

Rp	R1	R2	R3	R4	R5	R6	R7	R8	R9
10000	1100	1300	1800	2400	3300	5100	8200	16000	51000

Table B3 Voltage at the ADC input and 8-bit conversion result (5V supply)

Active key	Vmin	Vmax	Conversion result
Key0	0.00	0.00	0-0

Key1	0.48	0.51	24-26
Key2	0.94	1.00	48-51
Key3	1.44	1.52	73-78
Key4	1.94	2.04	99-104
Key5	2.44	2.54	124-129
Key6	2.95	3.05	151-156
Key7	3.45	3.54	179-180
Key8	3.95	4.02	202-205
Key9	4.48	4.52	229-230

The condition *no key pressed* corresponds to a result of 255.

B2.3 Software Generation

The functional description of the application includes:

- The analog input through an ADC to read the value issued by the voltage dividers.
- Key recognition.
- Transfer of the result to other functional blocks, or conditional jumps in a state machine.

The recognition of the pressed key is achieved by comparing the digitized analog value with the range limits defined in Table B3. However, these ranges [0-0], [24-26] up to [229,230] are not contiguous, thus two comparisons (upper limit and lower limit) are required to check that a value is within a range. This means that more ROM and RAM is used with an higher execution time. Thus some extended ranges with common limits are defined as shown in Table B4.

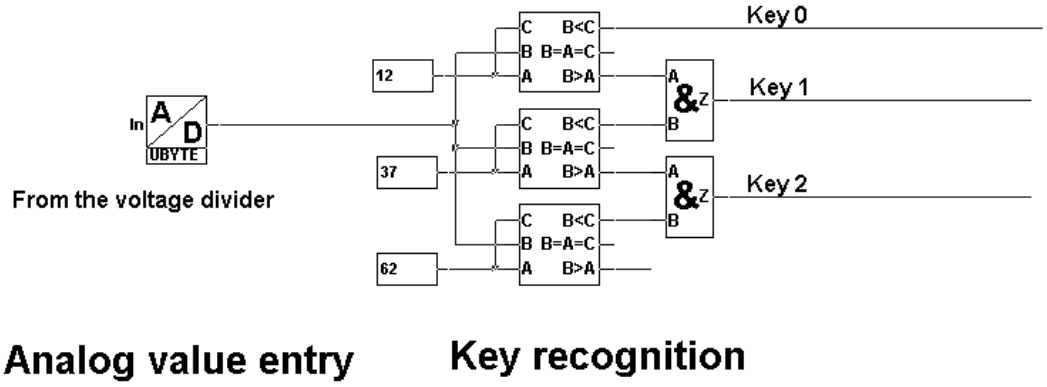
Table B4 Recognition ranges

Key pressed	Recognition ranges
Key0	0-12
Key1	12-37
Key2	37-62
Key3	62-88
Key4	88-114
Key5	114-140
Key6	140-165
Key7	165-190
Key8	190-217
Key9	217-244
None	244-255

A range limit value is never reached (see Table B3), therefore no ambiguous situation can occur.

The key pressed can then be recognized with numeric comparators and some logical gates, as shown in the Figure B8. The output signals generated *Key 0,.., Key 9, None* are logical signals, and can therefore be used both as input signals to other functions or as conditions in a state machine.

Figure B8 Key Recognition by Analog Value Evaluation



B2.4 Possible Improvements

The inputs A and C of the comparators are interpreted as variables to which fixed values are assigned: the range limits. Even though this does not have any importance for the application process itself, it must be taken into account in some cases. Some RAM locations, usually dedicated to variable storage, are used to store these constant values. This reduces the RAM space available for the surrounding application (Data processing, I/O control etc.).

This can be improved by creating a specific comparison symbol where the reference values are defined as constant. Thus the limit values are stored in the ROM and not in the RAM space.

While doing this, it is also possible to fine tune the symbol function for the application. The new symbol is defined as follows:

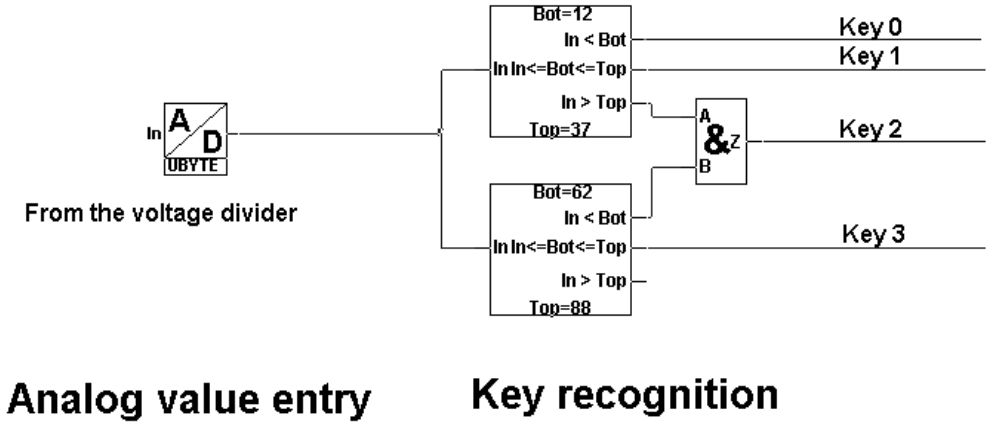
The variable input value is compared to 2 reference values *Bot, Top* attached to the symbol instance, providing 3 output:

- Input > Top
- Bot <= Input <= Top
- Input < Bot

Only 5 comparators and one RAM location are used by the digitized representation of the voltage value, instead of 11 in the previous case (input value plus 10 border values).

The symbol customisation feature provided by the ST-Realizer environment thus enables a more simple graphical description to be created (only 5 comparators) while optimizing the memory requirements (Figure B9).

Figure B9 System Optimization Using Customized Components



B3 Clock Design

MCU solutions are often used as clocks owing to the onboard timer and all the task management possibilities offered by the MCU embedded resources. This approach enables single-chip solutions to be developed that are extremely useful in applications such as small home appliances.

The following paragraphs show an example clock design, that includes an alarm feature, that was developed using ST-Realizer.

B3.1 Application Overview

The clock system provides the following features:

- Current time counting
- Alarm triggering at a defined time
- Current time setup
- Alarm time setup

The time values are represented in the HH:MM format, but the described concept can easily be extended to representation in seconds. By using this HH:MM format, the time value is represented by a pair of integer variables ranging between [0 to 59] for the minutes and [0 to 23] for the hours.

The user interface consists of 4 keys: TIME SETUP, ALARM SETUP, HOURS and MINUTES.

With these 4 keys, both the current time and the alarm time can be adjusted:

- When the key TIME SETUP is activated, the *Hours (Resp. Minutes)* variable of the current time is increased each time the key HOURS (Resp. MINUTES) is activated.
- When key ALARM SETUP is activated, the *Hours (Resp. Minutes)* variable of the Alarm time is increased each time the key HOURS (Resp. MINUTES) is activated.

All time variables are reset to 00 when they reach the maximum value (23 for the hours and 59 for the minutes).

B3.2 Current Time Counting

The system uses a timebase generated by the clock of the embedded timer on the ST6 or ST7 MCU. This timebase can be configured (the default value is 10 ms). It is used to trigger three chained modulo-n counters (Unit Counter):

- One for seconds (n = 60).
- One for minutes (n = 60).
- One for hours (n = 24).

When any of these Unit Counters reach their maximum value, a clock is issued to increment the Unit counter of the next stage.

B3.3 Current Time Setup

The current time value is modified by incrementation of the counters used for the current time counting. This is achieved by duplicating the clock input of the appropriate counter in the current time counting block. In practice, two different additional clocks are needed, one for minute incrementation and one for hour incrementation. Each of these additional clocks is controlled by the combination of the keys: TIME SETUP, HOURS and MINUTES.

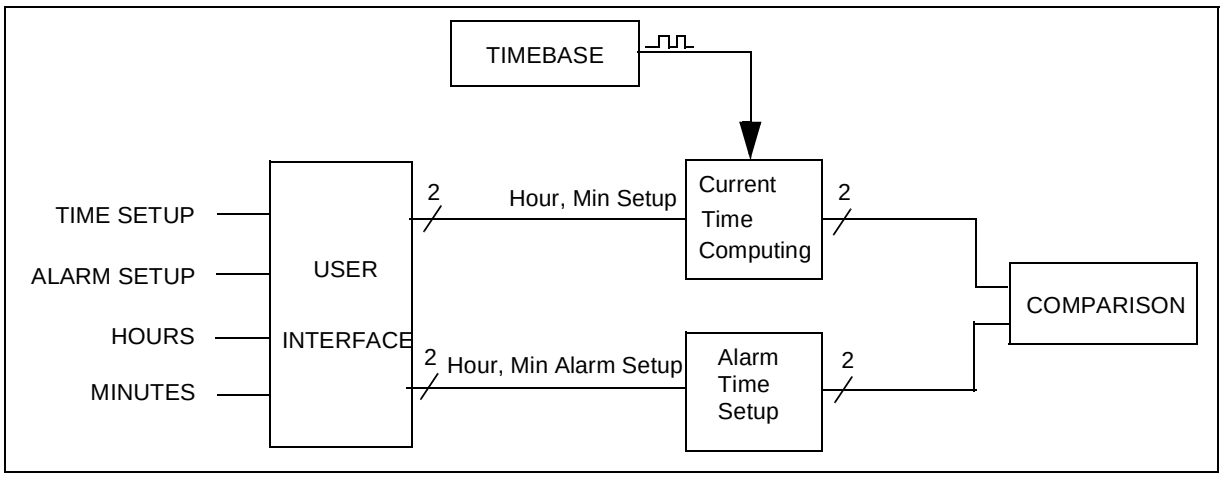
B3.4 Alarm Time Setup

This process is similar to current time setup. Two counters are used, one for minutes, and for hours. The contents of these counters can only be modified by pressing the keys ALARM SETUP, HOURS and MINUTES once.

B3.5 Alarm triggering

An alarm process is launched when the current time is equal to the predefined alarm time. The occurrence is enabled by a double comparison: Hours equal, Minutes equal

Figure B10 Application Block Diagram

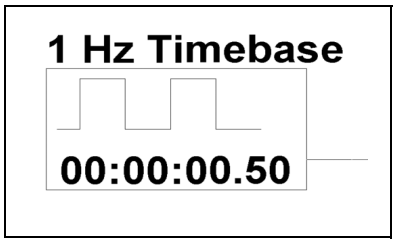


B3.6 Timebase

The timebase is described using the Oscillator component of the ST-Realizer library. Any “real-time” value can be defined for the period. For this application, the 1Hz timebase is defined as an oscillating square wave with half a period (level High duration) of 00:00:00.50 in the HH:MM:SS.xx format (Figure B11).

ST-Realizer uses the embedded Timer of the ST6 or ST7 MCU as the timebase, and generates by software any periodic variable.

Figure B11 Timebase Description



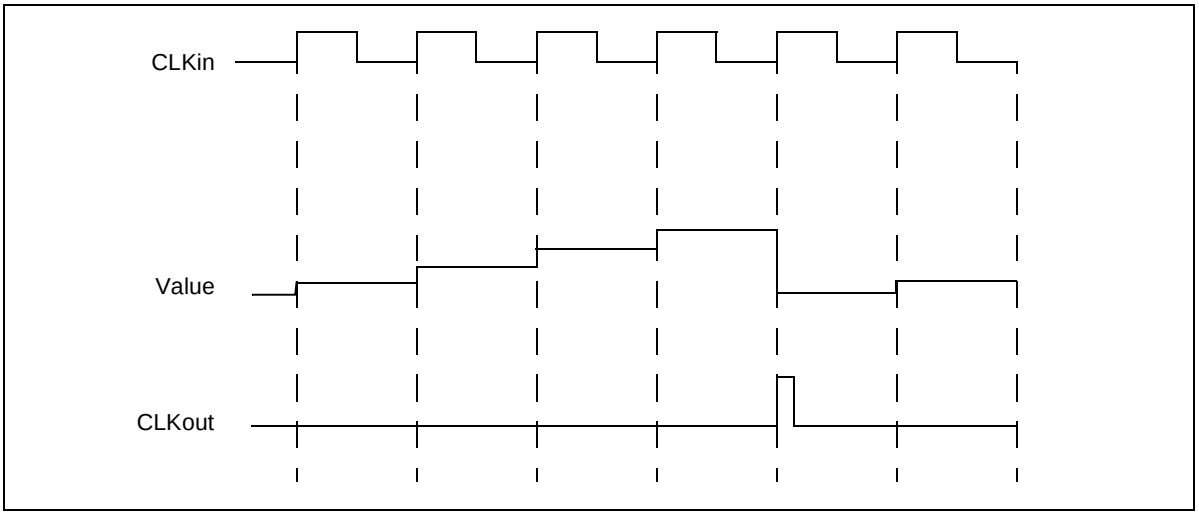
B3.7 Current Time Counting

Each of the 3 chained Modulo-N counters presents:

- An input clock issued from the previous stage (timebase for the 1st stage).
- An output clock to activate the following stage.
- A byte-wise output with the current counted value (Second, Minute or Hour).
- An optional clock for the incremental set-up process.

Any content change can only be performed during an input clock transition from 0 to 1. Thus the reset phase when the counter reaches its maximum value has to be anticipated. For example, the Hours Modulo-24 counter is reset if the two following conditions occur: contents equal 23 and input clock transition. In parallel, the resulting Reset signal is issued as output clock to increment the next stage as shown in Figure B12 in the case of a Modulo 5 counter.

Figure B12 Clock synchronism principle



Each Modulo-N block is based on a Counter component provided within the ST-Realizer standard library. This symbol presents the advantage of generating a numeric variable as

output. This means that any numeric or arithmetic operation can be directly performed on that variable without modifying its binary representation. The detection of the maximum value is achieved with a comparator symbol.

The comparator output is entered into the feedback loop at the falling edge of the input clock. This avoids any parasitic Reset during the High level of the input clock, when the comparator output switches to High level. In parallel, the counter incrementation is prevented by forcing the *Up* input of the counter to Low level.

Finally, each Unit Counter is represented in the ST-Realizer environment as shown in the Figure B13.

Figure B13 Unit Counter Description

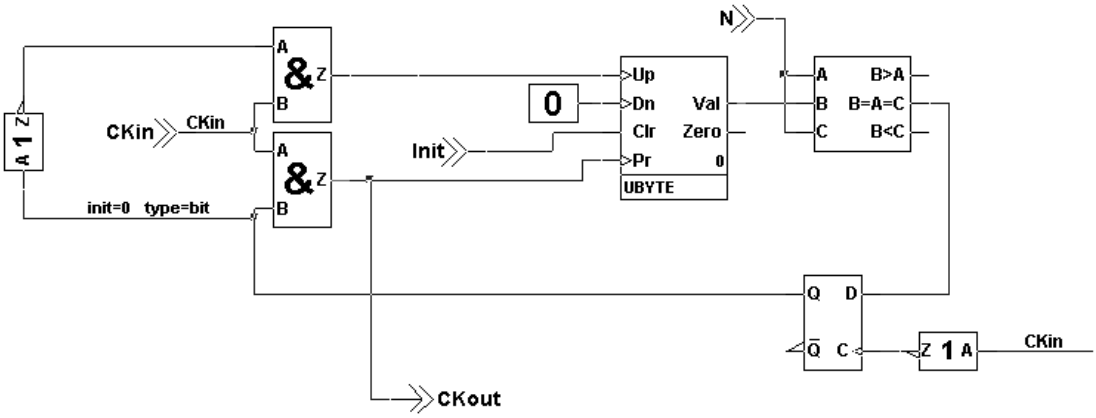
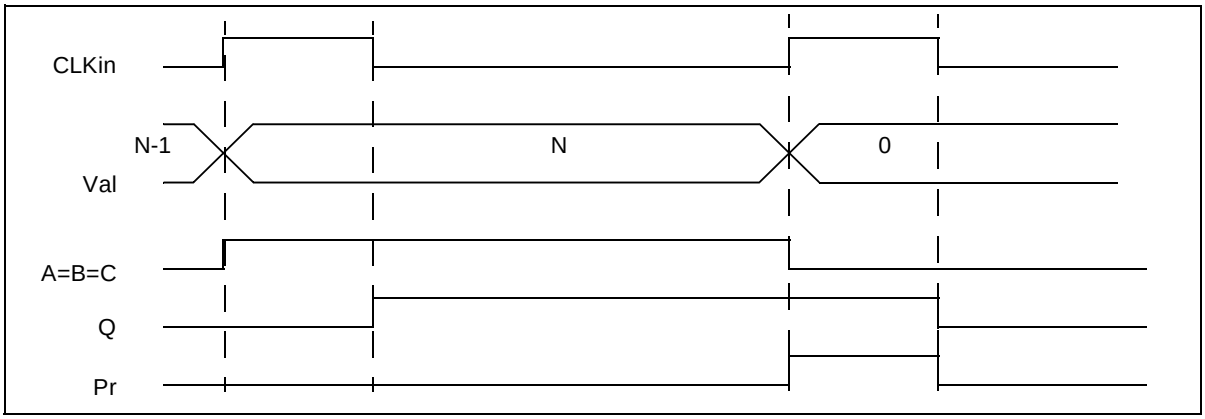


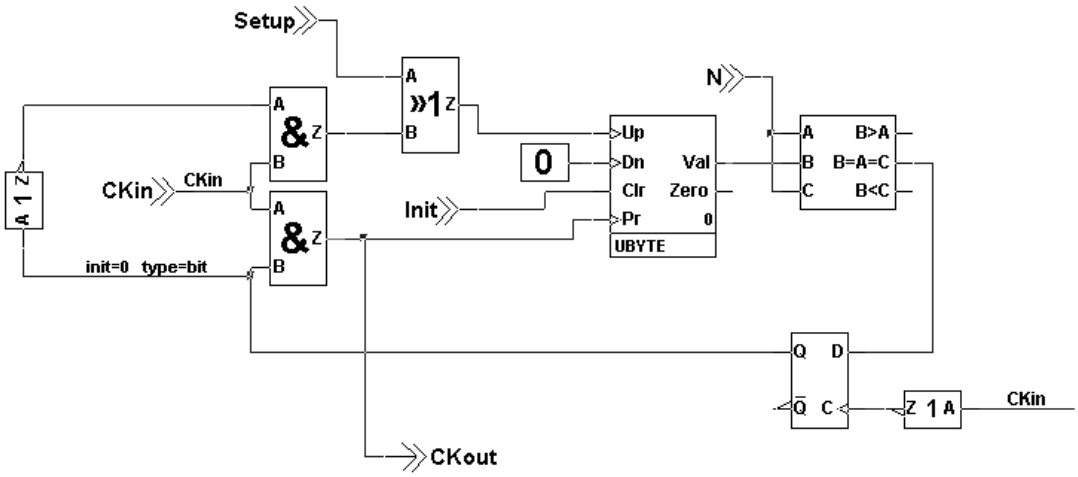
Figure B14 Current Time Counter Waveforms



B3.8 Current Time Setup

The incrementation clock (*Up* input of the symbol) is duplicated through an OR gate with an external clock, *Setup* signal, for Hour or Minute setup. Any activation on this clock increments the content of the counter by 1 (Figure B15).

Figure B15 Current Time Setup with a Duplicated Clock

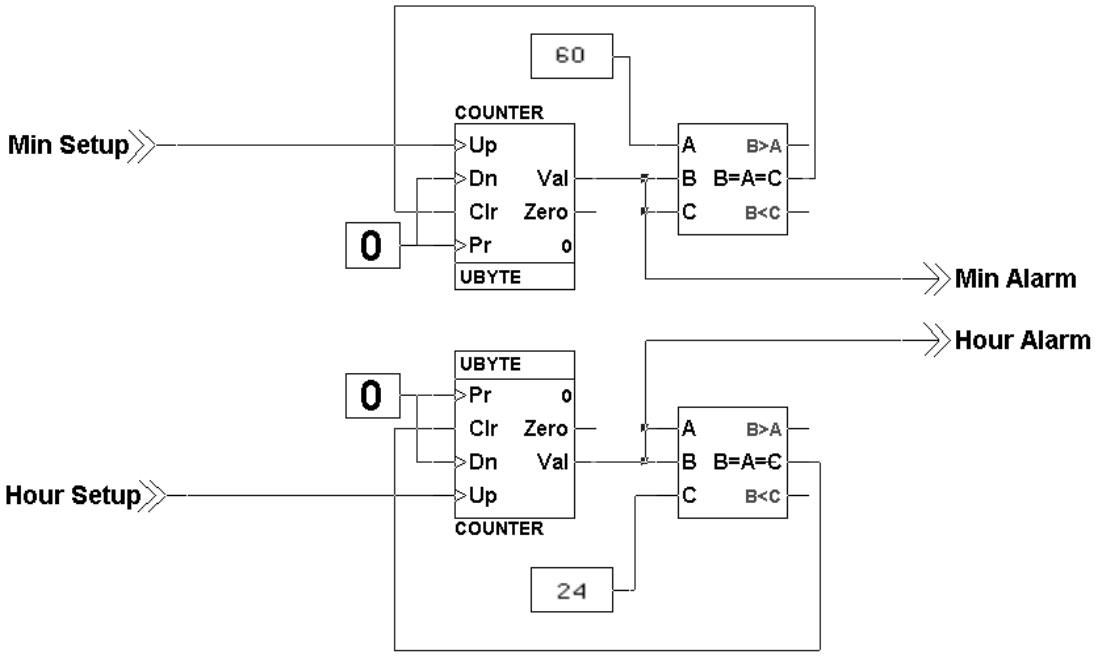


B3.9 Alarm Time Setup

As for the Current Time setup, the Alarm Time setup is achieved by incrementation of two Modulo-N counters, one for minutes and one for hours. Each of them has its own input clock, controlled by the combination of the keys ALARM SETUP, HOURS and MINUTES.

A feedback loop is still needed to reset the counter when the maximum value is reached, but its implementation can be more simple than in the Current Time counting blocks. A precise synchronism is not mandatory, and a basic solution can be used (Figure B16).

Figure B16 Alarm Time Setup

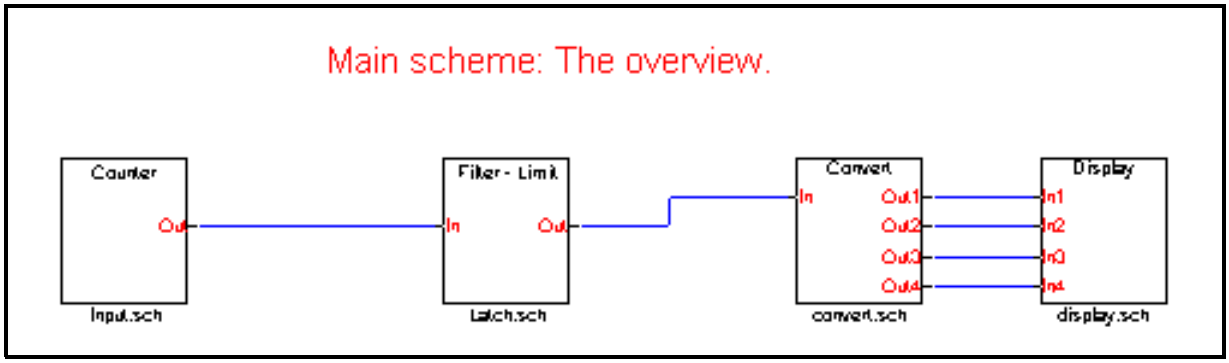


B4 Fast Counter Application

B4.1 The Application

The following description of an application is added to this appendix to illustrate the new features that come with ST-Realizer II. The application is a pulse counting device used to measure the frequency of a digital signal. It uses the interrupt sensitive inputs of an ST6260 microcontroller. It will filter the measured frequency so that a readable value is obtained. It converts these values into viewable data and then drives a 4-digit 7-segment LED-display. Figure B17 shows the basic block diagram symbols that represent the subschemes. The next diagrams show the 4 sub schemes in detail. Parts of these sub schemes are used in the first part of this application note. Please note that they are all made using ST-Realizer II. In Section B4.2, the Realizer Report File (FASTCNT.RPF) is shown. It informs the designer what resources are used in which way. In addition, the code generated by Realizer is shown, with some comments added for your convenience.

Figure B17 Overview of the Fast Counter Application



The subscheme shown in Figure B18 calculates the moving average over 4 samples every 3 seconds.

Figure B18 Fast Counter: Input section

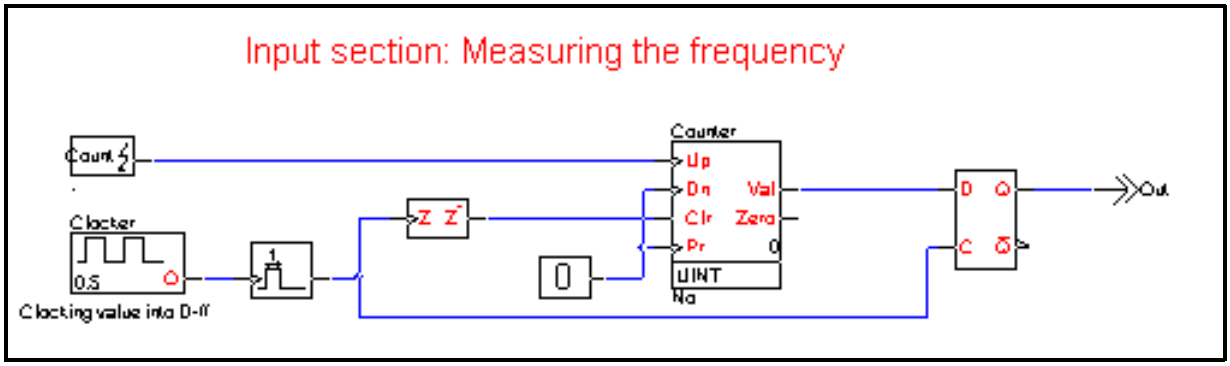
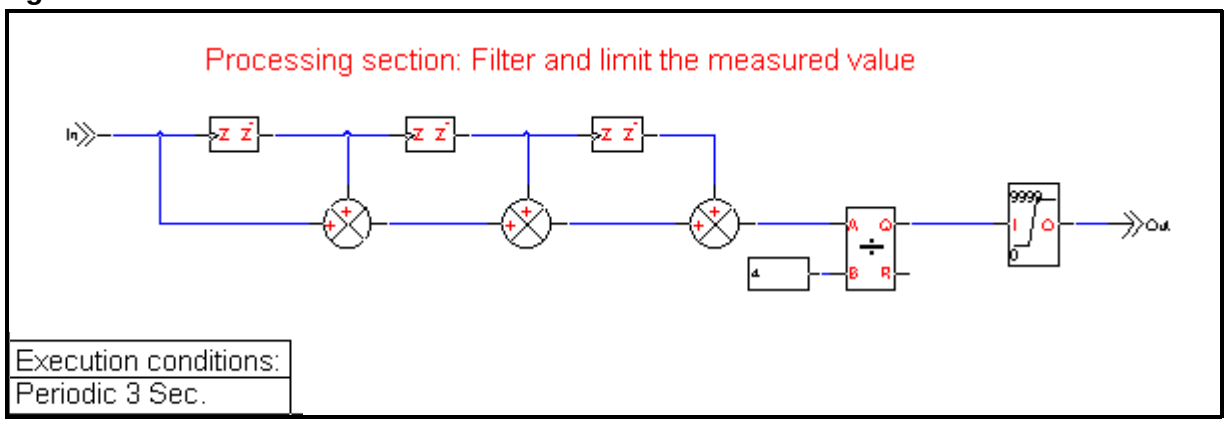


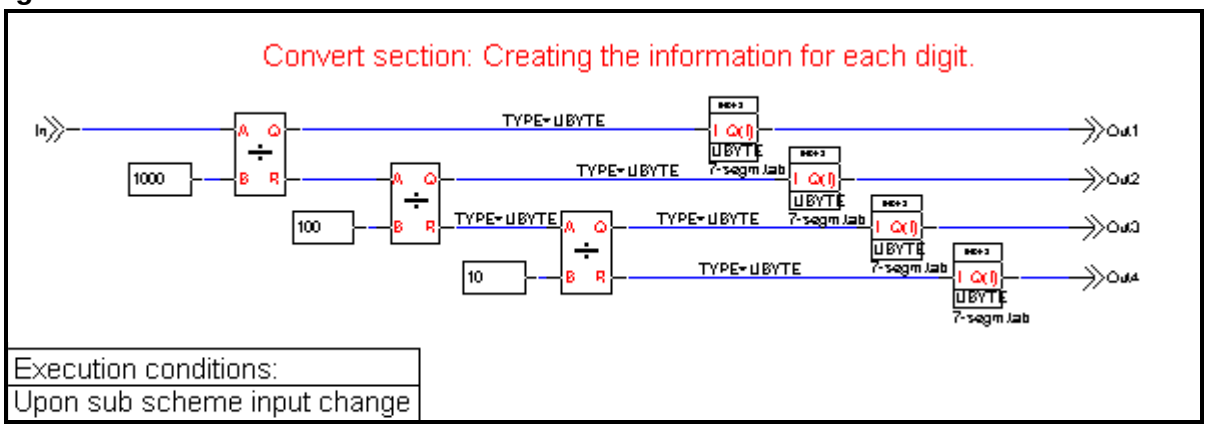
Figure B19 Fast Counter: Filter and Limit section



The subscheme shown in Figure B19 counts under interrupt. The counter value is latched every second. The edge detection symbol is needed to de-assert the Clr input during the high-time of the oscillator.

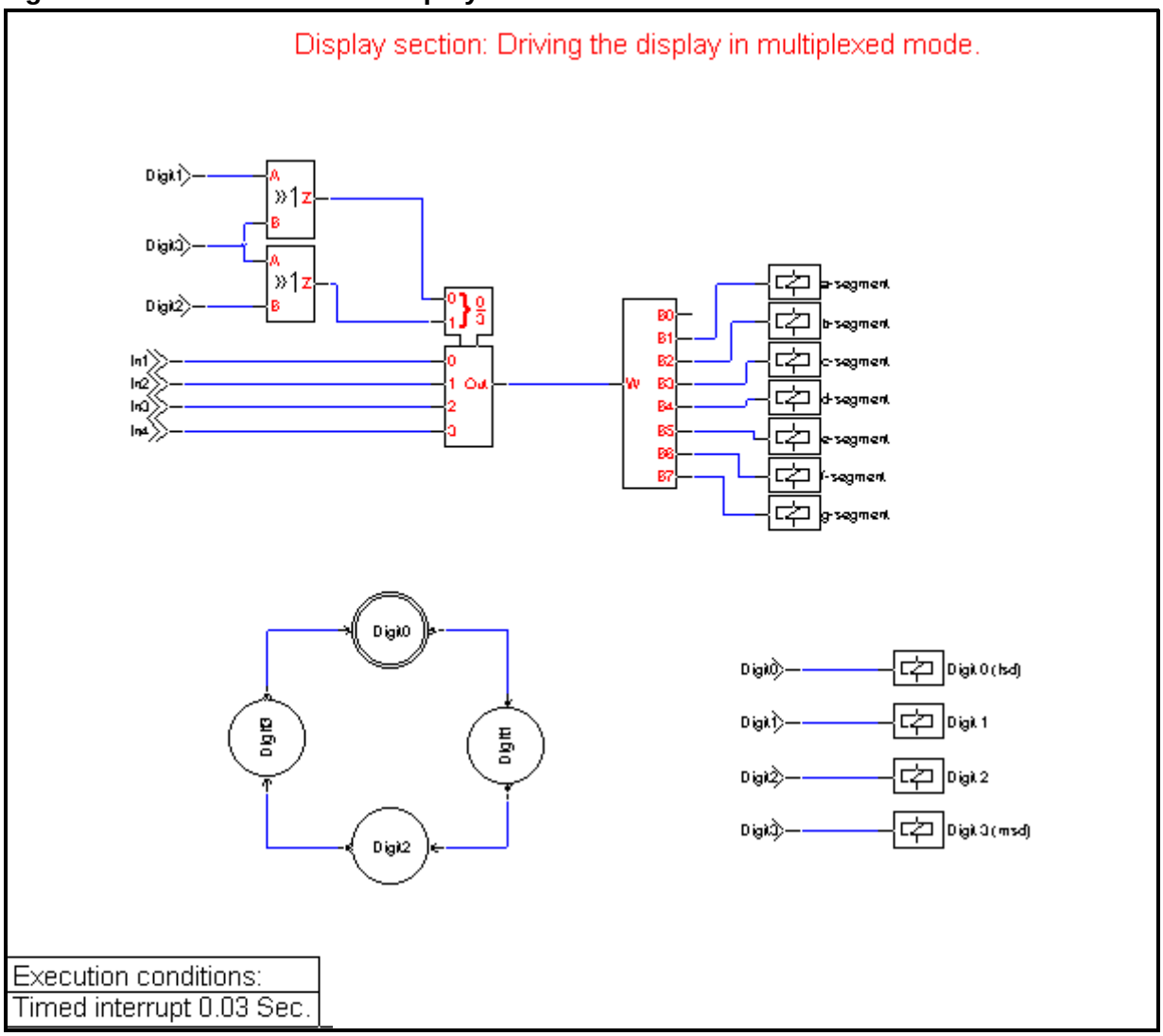
The subscheme shown in Figure B20 converts the integer value into 4 bytes that carry the data for the 7-segment displays. It is executed only when the input changes.

Figure B20 Fast Counter: Conversion section



The subscheme shown in Figure B21 takes care of the multiplexing of 4 7-segment displays. It runs under interrupt control, every 10 milliseconds.

Figure B21 Fast Counter: Display section



B4.2 Fast Counter Report File

Here is the Report File generated for the sample application discussed in this section.

```
------------------------------------------------------------------------
ST6260 Realizing Unit (V4.00) (c) 1990-99 Actum Solutions
Report file of project C:\Program Files\ST-
Realizer\Examples\Fastcnt\FastCnt.rpf
Scheme Version   : 1.04
Report timestamp : Tue May 18 14:06:28 1999
Analyze results  : No errors
------------------------------------------------------------------------

Schematic dependencies and events:
------------------------------------------------------------------------

C:\Program Files\ST-Realizer\Examples\Fastcnt\Fastcnt.sch
  Scheme: C:\Program Files\ST-Realizer\Examples\Fastcnt\Latch.sch
    Event:  Periodic 3 Sec.
  Scheme: C:\Program Files\ST-Realizer\Examples\Fastcnt\Input.sch
    Event:  Count=PB.2 interrupt
  Scheme: C:\Program Files\ST-Realizer\Examples\Fastcnt\convert.sch
    Event:  Upon sub scheme input change
  Scheme: C:\Program Files\ST-Realizer\Examples\Fastcnt\display.sch
    Event:  Timed interrupt 0.03 Sec.


ST6260 (DIL20) connection overview:
------------------------------------------------------------------------

Pin   Name  Alternative name   Type  I/O         Description
  1:  PB.0  Digit 0 (lsd)     (BIT  Output ),   20 mA sink open drain output
  2:  PB.1  Digit 1            (BIT Output ),   20 mA sink open drain output
  3:  TEST                    (          ),    Test/program
  4:  PB.2  Count             (BIT  Input ),    interrupt
  5:  PB.3                    (BIT  Input ),    Not connected
  6:  PB.6  Digit 2           (BIT  Output ),   20 mA sink open drain output
  7:  PB.7  Digit 3 (msd)     (BIT  Output ),   20 mA sink open drain output
  8:  PA.0  a-segment         (BIT  Output ),   5 mA sink open drain output
  9:  Vdd                     (          ),    Power supply
 10:  Vss                     (          ),    Power supply
 11:  PA.1  b-segment         (BIT  Output),    5 mA sink open drain output
 12:  PA.2  c-segment         (BIT  Output),    5 mA sink open drain output
 13:  PA.3  d-segment         (BIT  Output),    5 mA sink open drain output
 14:  OSCin                   (          ),    Oscillator
 15:  OSCout                  (          ),    Oscillator
 16:  RESET                   (BIT  Input ),    Active low
 17:  NMI                     (BIT  Input ),    Non Maskable Interrupt
 18:  PC.4  f-segment         (BIT  Output),    5 mA sink open drain output
 19:  PC.3  g-segment         (BIT  Output),    5 mA sink open drain output
 20:  PC.2  e-segment         (BIT  Output),    5 mA sink open drain output
```

Hardware connections:

Symbolic name	H/W name	Description	Comment
Digit 0 (lsd)	PB.0	20 mA sink open drain output	
Digit 1	PB.1	20 mA sink open drain output	
Digit 2	PB.6	20 mA sink open drain output	
Digit 3 (msd)	PB.7	20 mA sink open drain output	
a-segment	PA.0	5 mA sink open drain output	
b-segment	PA.1	5 mA sink open drain output	
c-segment	PA.2	5 mA sink open drain output	
d-segment	PA.3	5 mA sink open drain output	
e-segment	PC.2	5 mA sink open drain output	
f-segment	PC.4	5 mA sink open drain output	
g-segment	PC.3	5 mA sink open drain output	

Variable overview:

```
Total used bits              : 23
Total used events            : 5
Total used unsigned bytes    : 20
Total used signed bytes      : 0
Total used unsigned integers : 19
Total used signed integers   : 0
Total used longs             : 0
```

Memory overview:

```
Total used RAM               : 73 byte (000H->03FH,085H->08CH)
Total used ROM               : 1714 byte (0080H->0732H) of 0F9DH
```

Note: The pins that are not connected are defined as digital input with pull-up. The timer pin is configured as an output.

B4.3 Generated Code

Code generated for the sample application discussed in this section.

```
; ST Realizer (Beta 3) (V4.00) : generated ST6260 Code
; File           : C:\Program Files\ST-Realizer\Examples\Fastcnt\Fastcnt.asm
; Scheme Version : 1.04
; Date           : Tue May 18 14:06:27 1999
; Used variables : 52
; Used functions : 100


.VERS "ST6260"
.ROMSIZE 4
.DP_ON

.LIST 0
.INPUT "Fastcnt.inc"
.INPUT "C:\Program Files\ST-Realizer\lib\ST6260.inc"
.INPUT "C:\Program Files\ST-Realizer\lib\st62lib.mac"
.INPUT "C:\Program Files\ST-Realizer\lib\st62xx.mac"
.INPUT "C:\Program Files\ST-Realizer\lib\st62eepr.mac"
.LIST 1

.ORG 00080H

RROMST:

.ASCIZ "Fastcnt"
.ASCIZ "1.04"
Reset:
  LDI IOR,IOR_MASK
  LDI HWDR,0FFH

  LDI DRBR,010H
  LDI EECTL,000H

PerInit:

;  initialize PORTB events

;    Enable interrupt on PORTB.2
PortInit:
  LDI DDRA,00FH
;  LDI ORA,000H
  LDI DRA,00FH
  LDI BUDRA,00FH
  LDI DDRB,0C3H
  LDI ORB,004H
  LDI DRB,0C3H
  LDI BUDRB,0C3H
  LDI DDRC,01CH
;  LDI ORC,000H
  LDI DRC,01CH
```

```
  LDI BUDRC,01CH

Rtcinit:
  LDI PSC,015H
  LDI TCR,0CFH
  LDI TSCR,06DH
  CLR TICK
  LDI IOR,(IOR_MASK | 010H)
  RETI

RamInit:
  LDI A,0
  LDI X,000H
  LDI Y,62
RamInit1:
  LD (X),A
  INC X
  DEC Y
  JRNZ RamInit1

  iconstw v1n7,2,4
  iconstb v2n4,5,1,0
  idltchwbww v2n10,4,v2n11,2,1,y2n9,4,0,0
  iconstw v3n20,2,10
  iconstw v3n19,2,100
  iconstw v3n18,4,1000
  JP EIS03011
IS03011:
.BYTE  000H
.BYTE  07EH
.BYTE  001H
.BYTE  030H
.BYTE  002H
.BYTE  06DH
.BYTE  003H
.BYTE  079H
.BYTE  004H
.BYTE  033H
.BYTE  005H
.BYTE  05BH
.BYTE  006H
.BYTE  01FH
EIS03011:

RealMain:
Rtc:
  LDI IOR,IOR_MASK
  LD A,TICK
  CLR TICK
  LDI IOR,(IOR_MASK | 010H)
  LD RTICK,A
RTIMEND:

  LD A,RTICK
```

```
  JRNZ IRTC0003
  JP RTCSKIP
IRTC0003:
;  Decrement 16 bit timers
  JRS 7,037H,RTC0003
  sub2www 037H,4,RTICK,2,037H,4
RTC0003:

RTCSKIP:

RINPEND:

;   moved to TIMER1 interrupt:   CALL SUB0002
  LDI IOR,IOR_MASK
  copyww y2n9,4,v0n5,4
  LDI IOR,(IOR_MASK | 010H)

;   moved to PORTB interrupt:   CALL SUB0002

;  check for time-out
  JRR 7,AT00001,_UNI0000
  LDI AT00001+0,1
  LDI AT00001+1,43
  CALL SUB0001
_UNI0000:

;  Input change detection on v0n7
  LD A,v0n7+0
  CP A,pv0n7+0
  JRZ _UNI0001
  JP EXEC0003
_UNI0001:
  LD A,v0n7+1
  CP A,pv0n7+1
  JRZ _UNI0002
  JP EXEC0003
_UNI0002:
  JP NOEX0003
EXEC0003:
  CALL SUB0003
NOEX0003:

;  Disable interrupts for parameter passing
  LDI IOR,IOR_MASK
  copyww v0n11,2,x4n5,2
  copyww v0n8,2,x4n2,2
  copyww v0n10,2,x4n4,2
  copyww v0n9,2,x4n3,2

;  Enable interrupts
  LDI IOR,(IOR_MASK | 010H)

;   moved to TIMER1 interrupt:   CALL SUB0004
ROUTPEND:
```

```
  copyww v0n7,4,pv0n7,4
  LDI IOR,IOR_MASK
  LD A,BUDRA
  LD DRA,A
  LD A,BUDRB
  LD DRB,A
  LD A,BUDRC
  LD DRC,A
  LDI IOR,(IOR_MASK | 010H)
  LDI HWDR,0FFH
  JP RealMain

SUB0001:
  loopdelwww v0n5,4,sv1n2,4,v1n0,4
  loopdelwww v1n0,4,pv1n0,4,v1n1,4
  loopdelwww v1n1,4,pv1n1,4,v1n5,4
  add2www v0n5,4,v1n0,4,v1n3,4
  add2www v1n3,4,v1n1,4,v1n4,4
  add2www v1n4,4,v1n5,4,v1n6,4
  divwwww v1n6,4,v1n7,2,v1n9,4,0,0
  limfww v1n9,4,v0n7,4,0,9999
  copyww v0n5,4,sv1n2,4
  copyww v1n0,4,pv1n0,4
  copyww v1n1,4,pv1n1,4
  ret

SUB0002:    ; interrupt driven sub routine
  oscfb v2n13,0,1,49,100,T02006,2
  edgebbb v2n13,0,1,pv2n13,1,1,v2n11,2,1
  loopdelbbb v2n11,2,1,pv2n11,3,1,v2n3,4,1
  eventb v2n12,6,1
  countfbbbbbbbwb
v2n12,6,1,pv2n12,7,1,v2n4,5,1,pv2n4,0,1,v2n3,4,1,v2n4,5,1,pv2n4,0,1,v2n10,4,
0,0,0,0,CT02000,4
  dltchwbww v2n10,4,v2n11,2,1,y2n9,4,0,0
  copybb v2n13,0,1,pv2n13,1,1
  copybb v2n11,2,1,pv2n11,3,1
  copybb v2n4,5,1,pv2n4,0,1
  ret   ; end of interrupt driven sub routine

SUB0003:
  divwwww v0n7,4,v3n18,4,v3n16,2,v3n10,4
  divwwww v3n10,4,v3n19,2,v3n27,2,v3n12,2
  divwwww v3n12,2,v3n20,2,v3n26,2,v3n21,2
  indtabww v3n21,2,v0n11,2,S03011,14,0
  indtabww v3n26,2,v0n10,2,S03011,14,0
  indtabww v3n27,2,v0n9,2,S03011,14,0
  indtabww v3n16,2,v0n8,2,S03011,14,0
  ret

SUB0004:    ; interrupt driven sub routine
  stateoutb v4n22,1,1,st0,2,2
  stateoutb v4n21,2,1,st0,2,3
  or2bbb v4n21,2,1,v4n22,1,1,v4n1,3,1
  stateoutb v4n19,4,1,st0,2,1
```

```
  or2bbb v4n19,4,1,v4n21,2,1,v4n0,5,1
  mux2bbwwwww v4n0,5,1,v4n1,3,1,x4n2,2,x4n3,2,x4n4,2,x4n5,2,v4n6,2
  bunpackwbbbbbbbb
v4n6,2,0,0,0,v4n43,6,1,v4n47,7,1,v4n42,0,1,v4n41,1,1,v4n44,2,1,v4n45,3,1,v4n
46,4,1
  digoutb v4n46,4,1,BUDRC,3,1
  digoutb v4n45,3,1,BUDRC,4,1
  digoutb v4n44,2,1,BUDRC,2,1
  digoutb v4n42,0,1,BUDRA,2,1
  digoutb v4n47,7,1,BUDRA,1,1
  digoutb v4n43,6,1,BUDRA,0,1
  digoutb v4n41,1,1,BUDRA,3,1
  stateoutb v4n12,5,1,st0,2,0
  digoutb v4n12,5,1,BUDRB,0,1
  stateoutb v4n13,6,1,st0,2,1
  digoutb v4n13,6,1,BUDRB,1,1
  stateoutb v4n14,7,1,st0,2,2
  digoutb v4n14,7,1,BUDRB,6,1
  stateoutb v4n15,0,1,st0,2,3
  digoutb v4n15,0,1,BUDRB,7,1
  stateminit st0,2,0
  stateminit st0,2,1
  stateminit st0,2,2
  stateminit st0,2,3
  stateminit st0,2,-1
  stateinit st0,2,0
  state st0,2,0,1
  stateend st0,2,0
  stateinit st0,2,1
  state st0,2,1,2
  stateend st0,2,1
  stateinit st0,2,2
  state st0,2,2,3
  stateend st0,2,2
  stateinit st0,2,3
  state st0,2,3,0
  stateend st0,2,3
  statemend st0,2,4
  LD A,BUDRA
  LD DRA,A
  LD A,BUDRB
  LD DRB,A
  LD A,BUDRC
  LD DRC,A
  ret   ; end of interrupt driven sub routine

.IFC NDF Rtcint
Rtcint:
  LDI TSCR,000H
  LDI PSC,015H
  LDI TCR,0CFH
  LDI TSCR,06DH
  INC TICK

;  Create normal stack push
```

```
  LD STACKA,A
  LD A,X
  LD STACKX,A
  LD A,Y
  LD STACKY,A
  LD A,V
  LD STACKV,A
  LD A,W
  LD STACKW,A
  LD A,REG0
  LD STACK0,A

;  Decrement interrupt timers
  JRS 7,T02006,TM1_0004
  decw T02006,2
  JRR 7,T02006,TM1_0004
  CALL SUB0002
TM1_0004:

;  Decrement prioritized timers
  decw AT00004,2
  JRR 7,AT00004,TM1_0005
  CALL SUB0004
  copyww 2,TCONST,AT00004,2
TM1_0005:

;  Create normal stack pop
  LD A,STACK0
  LD REG0,A
  LD A,STACKW
  LD W,A
  LD A,STACKV
  LD V,A
  LD A,STACKY
  LD Y,A
  LD A,STACKX
  LD X,A
  LD A,STACKA
  RETI
.ENDC


;  PORTA,PORTB interrupt service routine
.IFC NDF PORTAint
PORTAint:

;  Create normal stack push
  LD STACKA,A
  LD A,X
  LD STACKX,A
  LD A,Y
  LD STACKY,A
  LD A,V
  LD STACKV,A
```

```
  LD A,W
  LD STACKW,A
  LD A,REG0
  LD STACK0,A
  SET 6,v2n12
  CALL SUB0002
  RES 6,v2n12

;  Create normal stack pop
  LD A,STACK0
  LD REG0,A
  LD A,STACKW
  LD W,A
  LD A,STACKV
  LD V,A
  LD A,STACKY
  LD Y,A
  LD A,STACKX
  LD X,A
  LD A,STACKA
  RETI
.ENDC

RROMEND:

.ORG 0FFEH
  JP Reset

.ORG 0F9EH
  JP Reset

.ORG 0FF0H
  JP Rtcint

.ORG 0FFCH
  NOP
  RETI

.ORG 0FF6H
  JP PORTAint

.ORG 0FF4H
  NOP
  RETI

.ORG 0FF2H
  NOP
  RETI
```

Symbols

attribute 206
.asm file 129
.hex file 11, 129
.ini files 56
.log file 160
.obj file 11, 129
.rpf file 55
.s19 file 11, 129
.sch file 67
.sef files
  creating 139
  opening 141
  saving 142
.wmf file 67, 142

A

ACTUM solutions
  internet address 3
adc symbol 80
Adjusters 142
  numeric 143
    attaching 143
    options 145
    setting values 144
  sine wave 148
    placing 148
  square wave 150
    placing 150
    setting values 150
  time table 146
    placing 146
    setting values 146
  values
    recording 139, 160
    reusing 139, 160
ALLOCATEBLOCKIN attribute 207
ALLOCATION attribute 207
Analog-to-digital symbol 80
Analyse
  execution 133
  what to do if there are errors 134
Analyse and compile report 135
Analyse error messages 134
  viewing and tracing 135
Analysing and generating your application 129
Application development steps 11
Applications
  introduction to 5
  simulation of 139
  structure of 4
Apxd variable
  Variables
    Apxd 64
Arguments for macros 188
Attaching numeric adjusters to pins 143
Attribute preference settings for symbols . 76
Attributes
  # 206
  ALLOCATEBLOCKIN 207
  ALLOCATION 207
  attaching to symbols 182
  BLOCKSIZEINUNITS 207
  CODE 207
  COMMENT 207
  DEVICE 207
  ICODE 208
  LABEL 208
  NAME 208
  OCODE 208
  SCHEME 208
  TABLE 208
  TIME 209
  TXT 209
  TYPE 206
  UNITTYPE 209
  VALUE 209
  viewing hidden 83
Automatic wiring 79

B

Backup file
  creating 194
Base Clock
  changing the timer tick value 131
BIT 203

BLOCKSIZEINUNITS attribute 207
BUDRx variable
Variables
BUDRx 64
Buttons in toolbar 199

C

Changing table data format 100
Changing the Target Microcontroller 61
Clock input pin 181
CODE attribute 207
Code generation
example of 230
options 130
structure 4
Colors of boxes 194
COMMENT attribute 207
Compiling
changing options 129
execution 133
Configuring events 95
Connecting
Application to Target Device
introduction 11
I/O symbols 80
Constant symbol
changing value 76
Creating
a new project 55
backup file 194
macros 187
guidelines 187
new schemes 67
projects 55
simulation environment file 139
your own symbol 165
Customization of ST-Realizer 193

D

Default clock 10
Description of ST-Realizer Events 85
DEVICE attribute 207
digin symbol 80
Digital input symbol 80
digout symbol 80
Displaying simulation information 158
Drawings
exporting 142
in symbol shapes 178

E

Earlier versions of Realizer
opening projects from 56
Editing table data 100
EEPROM 65
Environment options 193
Error messages 129
eventenable symbol 80
Events 85
configuring 95
description of 85
input interrupt 97
introduction to 8
introduction to event symbols 9
introduction to execution conditions 8
NMI signal 96
periodic 87, 95
peripheral interrupts 89
peripheral-specific 97
scheduled 87
subscheme input change 95
symbol 80
symbols 9, 85, 89
target-dependent 96
timed interrupts 88
upon subscheme input change 86
Execution Conditions 8, 85
Exporting drawings 142

F

Files
.asm 129
.hex 11, 129
.ini 56
.log 160
.obj 11, 129

.rpf 55
.s19 ..... 11, 129
.sch 67
.sef 139
.wmf 67, 142
importing into tables 101
log 139
ST macro-assembler language 129
Folders
created by ST-Realizer 2
Fonts for printer 83

G

Generated code (example) 230
Generated report file (example) 228
Ghost box colors 194

H

Hardware (see Target Hardware)
heating control system tutorial 13

I

ICODE attribute 208
Importing files into tables 101
Initialising the simulation 158
Input
interrupt (event) 97
pin 181
simulated values 142
symbols 80
variables used by ST-Realizer 64
inputlatch symbol 80
Installation
folders created 2

L

LABEL attribute 208
Libraries
adding new subscheme symbol to ... 169
adding new user-defined symbol to . 191
local 73
placing a symbol from 73
symbol 73
log file 139
LONG 203

M

Macro files 64
Macro parameters 188
Macros 208
arguments 188
creating 187
naming 188
Main symbol library 103
main.lib library 73
mainper.lib library 73
Memory Configuration 65
Memory configuration 62
Monitoring
simulated signals 151
Multitype pins 203

N

NAME attribute 208
Naming macros 188
NMI signal (event) 96
Non-volatile variables 207
Numeric adjusters 143

O

OCODE attribute 208
Opening
projects 56
schemes 69
simulation environment file 141
Optimisations 133
Oscilloscope probes 153
Output
pin 181
symbols 80
Output symbol 80
outputlatch symbol 80

P

Page preferences 196
Parameters for macros 188
Passive output pin 181
Periodic (event) 95
Peripherals
    enabling 66
    settings 62
Pin type 206
Pins
    multitype 203
Placing symbols in a scheme 72
portin symbol
    portout symbol 92
PortInit subroutine 64
Preference settings
    symbol attributes 76
Preferences
    pages 196
    printing 197
    screen 194
    symbols 198
    toolbar 199
Printer
    fonts 83
    setting up 83
Printing preferences 197
Probes
    numeric 152
    oscilloscope 153
        attaching 153
        changing properties of 154
    simulation 151
    state machine 156
    state machine probes, attaching 156
    values
        recording 139
        reusing 139
    values, recording 160
    values, reusing 160
Processing Cycle
    controlling the time of 131
Project files 12
    closing 57
    creating new 55
    opening 56
    opening projects from earlier versions of Realizer 56
    saving 57

R

RAM 65
RamInit subroutine 64
RealInit subroutine 64
Realizer Operating System 63
RealMain subroutine 64
Recording
    adjuster values 139, 160
    probe values 139, 160
    simulation information 158
Report
    analyse and compile 135
    file (example) 228
    printing of 138
Requirements
    hardware 1
    software 1
Reusing
    adjuster values 139, 160
    probe values 139, 160
ROM 65
Root scheme
    introduction to 7
root_directory 2
ROS 63
RTICK variable
    Variables
        RTICK 64
Running the simulation 158

S

Saving
    projects 57
    schemes 69
    simulation environment file 142
SBYTE 203

Scheduled (event) 87
SCHEME attribute 208
Schemes 7
  analysing 129
  creating 67
  creating subschemes 68
  introduction to 7
  opening 69
  placing symbols 72
  placing titles in 84
  printing 83
  root scheme 7
  saving 69
  simulating 139
  subschemes 7
  viewing options 82
  working in 82
Screen Preferences 194
Setting
  numeric adjuster options 145
  numeric adjuster values 144
  square wave adjuster values 150
  time table adjuster values 146
  up printer 83
Simulated signals
  monitoring 151
  viewing 151
Simulating your application 139
Simulation
  adjusters 142
  displaying information 158
  environment file
    creating 139
    opening 141
    saving 142
  initialising 158
  input values 142
  probes 151
  recording 158
  recording and reusing adjuster and probe values 160
  recording information 158
  run options 158
  running 158
  starting and stopping 158
Sine wave adjusters 148
SINT 203
Software requirements 1
Specifying the Target Hardware Device ... 59
Square wave adjusters 150
sssp_q symbol 92
ST6 59
ST6 users
  note to 59
ST7 59
ST-Analyser 129
  error messages 129
State history list 157
State machine 156
State machine probes
  attaching 156
Steps in application development 11
STMicroelectronics
  internet address 3
Stopping the simulation 158
ST-Realizer
  application structures 4
  subroutines 64
ST-Simulator 139
  running the 157
Subscheme input change (event) 95
Subschemes 7, 85, 92
  assigning execution conditions to 93
  connecting to root schemes 92
  creating 68, 92
  disconnecting execution conditions from 97
  introduction to 7
  leaving 98
  opening 93
  placing event symbol in 94
  symbols 92
    defining new 166
Symbol editor 165
  adding graphics 178
  adding pins to a symbol 181
  assigning attributesl 182
  changing graphic properties 179
  editing new symbols 176
  editing pin attributes 177

editing symbol information 180
generating macro headers 175
modifying attributes 185
opening 165
Symbol libraries 73
Symbol preferences 198
Symbols
adc 80, 103
add2 118
Analog-to-digital 80
and2 107
and3 107
and4 108
and6 108
and8 108
attaching attributes 182
attributes
editing 75
average 118
bpack 121
bunpack 122
change 108
comp 122
condition 127
connecting to target hardware device 80
constant 126
constb 126
constw 126
conversion 121
convert 122
counter 120
countf, pcountf 120
countv, pcountv 121
creating your own 165
defining a new subscheme symbol .. 166
defining new user-defined symbols .. 171
deleting 75
delf 116
delfoff 116
delfon 116
delv 116
delvo 117
delvoff 117
dff, pdff 109
dff-clr 109
differential 118
digin 80, 103
digout 80, 103
div 118
dlatch 109
drawing shapes in 178
edge 115
event 80, 104
eventenable 80, 104
hierarchical sheet 128
indextable 124
information
editing 180
viewing 78
input 80, 104
input and output 103
inputlatch 80, 104
inputsequence 106
integral 119
introduction to 6
inv 110
jkff 110
limf 119
limv 119
logic 107
lookuptable 124
loopdel 110
main symbol library 103
mathematical 118
microdelf 105
microdelv 106
mirroring 77
moving 75
mul 119
mux1 110
mux2 111
nand2 111
nand3 111
nand4 111
nand6 112
nand8 112
nor2 112
nor3 113
nor4 113
nor6 113

nor8 114
or2 114
or3 114
or4 114
or6 115
or8 115
oscf 117
oscv 117
Output 80
output 80, 104
outputlatch 80
outputsequence 106
persistent library symbols 103
pitable 125
placing in a scheme 72
portin 92, 128
portout 92, 128
power management 125
programming with 4
ramtable (volatile RAM) 125
rotating 77
scalert 119
scalerv 120
sequential 105
shift, pshift 122
slow 125
srff, psrff 115
sssp_q 92
sssx_y 128
state 127
state machine 6, 126
statein 127
stateinit, pstateinit 126
stateout 127
stop 125
sub2 120
table 124
table symbols 99
time related 116
timf 117
timv 118
title 128
user-defined 64
wait 126
waitsequence 107
wiring 78
wiring together 78
wmerge 123
wpack 123
wsplit 123
wunpack 124
xor 109

T

TABLE attribute 208
Table symbols
changing data format 100
editing data 100
importing data files into 101
inserting 100
Tables
data format
changing 100
editing data in 100
importing files 101
index 99, 208
lookup 99, 208
ROM 208
Target Hardware
changing the 61
configuring the 62
requirements 1
settings 62
specifying device 59
Target Microcontroller
(see Target Hardware)
Target-dependent events 96
TIME attribute 209
Time table adjusters 146
Timer ticks
changing the value of 131
introduction to 10
Timing 10
options 131
Toolbar buttons 199
Tutorial 13
TXT attribute 209
Type
inheritance (variables) 204

of a pin 206
overruling (variables) 205
TYPE attribute 206

U

UBYTE 203
UINT 203
UNITTYPE attribute 209

V

VALUE attribute 209
Value of a constant symbol 76
Variables
BIT 203
LONG 203
non-volatile 207
SBYTE 203
SINT 203
type inheritance 204
type overruling 205
UBYTE 203
UINT 203
WORD 203
View
hidden attributes 83
Viewing
simulated signals 151
symbol information 78

W

Wires
changing attributes 80
copying 79
deleting 79
drawing 78
mirroring 79
moving 79
pasting 79
rotating 80
Wiring
symbols together 78
automatically 79
WORD 203

Z

Zoom view 82

SALES OFFICES

EUROPE

DENMARK

2730 HERLEV
Herlev Torv, 4
Tel.: (45-44) 94.85.33
Telefax: (45-44) 948694

FINLAND

LOHJA SF-08150
Ratakatu, 26
Tel.: (358-19) 32821
Telefax.: (358-19) 3155.66

FRANCE

94253 GENTILLY Cedex
7 - avenue Gallieni - BP. 93
Tel.: (33-1) 47.40.75.75
Telefax: (33-1) 47.40.79.10

67000 STRASBOURG
20, Place des Halles
Tel.: (33-3) 88.75.50.66
Telefax: (33-3) 88.22.29.32

GERMANY

D-85630 GRASBRUNN
Bretonischer Ring 4
Postfach 1122
Tel.: (49-89) 460060
Telefax: (49-89) 4605454

D-30916 ISERNHAGEN
Ernst-Grote-Strasse 23A
Tel. (49-511) 9027513
Telefax: (49-511) 90275555

D-90491 NÜRNBERG 20
Erlenstegenstrasse, 72
Tel.: (49-911) 959840
Telefax: (49-911) 9598499

D-70499 STUTTGART 31
Mittlerer Pfad 2-4
Tel.: (49-711) 13968-0
Telefax: (49-711) 8661427

ITALY

20090 ASSAGO (MI)
V.le Milanofiori - Strada 4 - Palazzo A/4/A
Tel.: (39-2) 57546.1 (10 linee)
Telefax: (39-2) 8250449

40033 CASALECCHIO DI RENO (BO)
Via R. Fucini, 12
Tel.: (39-51) 591914
Telefax: (39-51) 591305

00161 ROMA
Via A. Torlonia, 15
Tel.: (39-6) 44251142
Telefax: (39-6) 85354438

NETHERLANDS

5652 AR EINDHOVEN
Meerenakkerweg 1
Tel.: (31-40) 2509600
Telefax: (31-40) 2528835

POLAND

WARSAW 00517
Ul. Marszalkowska 82
Tel.(0048-22) 622 0561
Telefax: (0048-22) 623 6437

SPAIN

E-08004 BARCELONA
Calle Gran Via Corts Catalanes, 322
6th Floor, 2th Door
Tel.: (34-3) 4251800
Telefax: (34-3) 4253674

E-28027 MADRID
Calle Albacete, 5
Tel. (34-1) 4051615
Telefax: (34-1) 4031134

SWEDEN

S-16421 KISTA
Borgarfjordsgatan, 13 - Box 1094
Tel.: (46-8) 7936920
Telefax: (46-8) 7504950

SWITZERLAND

1215 GENEVA 15
Route de Pré-Bois, 20
Tel.: (41-22) 9292929
Telefax: (41-22) 9292900

UNITED KINGDOM and EIRE

MARLOW, BUCKS
Planar House, Parkway
Globe Park
Tel.: (44-1628) 890800
Telefax: (44-1628) 890391

Internet Address http://www.st.com (Home Page)
Product Support http://www.st.com/stonline/products/support/index.htm

AMERICAS

BRAZIL

05413 SÃO PAULO
R. Henrique Schaumann 286-CJ33
Tel.: (55-11) 883-5455
Telefax : (55-11) 282-2367

CANADA

NEPEAN ONTARIO K2H 9C4
301 Moodie Drive Suite 307
Tel.: (613) 829-9944
Telefax : (613) 829-8996

U.S.A.

NORTH & SOUTH AMERICAN MARKETING HEADQUARTERS
55 Old Bedford Road
Lincoln, MA 01773
Tel.: (617) 259-0300
Telefax : (617) 259-4421

ALABAMA
Huntsville - Tel.: (205) 895-9544
Fax : (205) 895-9114

ARIZONA
Phoenix - Tel.: (602) 485-6201
Fax: (602) 485-6330

CALIFORNIA
Agoura Hills Tel. (818) 865-6850
Fax: (818) 865-6861
Santa Ana Tel. (714) 957-6018
Fax: (714) 957-3281
San Jose Tel.: (408) 452-8585
Fax: (408) 452-1549
Scotts Valley Tel.: (408) 439-2950
Fax: (408) 439-2969

COLORADO
Longmont Tel.: (303) 772-9729
Fax: (303) 772-0790

FLORIDA
Boca Raton Tel.: (561) 997-7233
Fax: (561) 997-7554

GEORGIA
Norcross Tel.: (770) 449-4610
Fax: (770) 449-4609

ILLINOIS
Schaumburg Tel.: (847) 517-1890
Fax: (847) 517-1899

INDIANA
Kokomo Tel.: (317) 455-3500
Fax: (317) 455-3400
Indianapolis Tel.: (317) 575-5520
Fax: (317) 575-8211

MICHIGAN
Livonia Tel.: (313) 953-1700
Fax: (313) 462-4071

MINNESOTA
Mineapolis Tel.: (612) 835-3500
Fax: (612) 835-3555

NORTH CAROLINA
Cary Tel.: (919) 469-1311
Fax: (919) 469-4515

NEW JERSEY
Voorhees Tel.: (609) 772-6222
Fax: (609) 772-6037
Basking Ridge Tel.: (908) 766-7401
Fax: (908) 766-7738

NEW YORK
Poughkeepsie Tel.: (914) 896-2926
Fax: (914) 897-3734

OREGON
Lake Oswego - Tel.: (503) 635-7635
Fax: (503) 635-7677

TENNESSEE
Knoxville - Tel.: (615) 524-6239
Fax: (615) 524-6247

TEXAS
Carrollton - Tel.: (972) 466-8844
Fax: (972) 466-8130
Houston - Tel.: (281) 376-9939
Fax: (281) 376-9948

FOR RF AND MICROWAVE POWER TRANSISTORS CONTACT THE FOLLOWING REGIONAL OFFICE IN THE U.S.A.

PENNSYLVANIA
Montgomeryville - Tel.: (215) 361-6400
Fax: (215) 361-1293

ASIA / PACIFIC

AUSTRALIA

DINGLEY VIC 3172
Suite 6, Business Centre
14 Garden Boulevard
Tel.: (61-3) 9558 3399
Telefax: (61-3) 9558 3377

NSW 2220 HURTSVILLE
Suite 3, Level 7, Otis House
43 Bridge Street
Tel.: (61-2) 9580-3811
Telefax: (61-2) 9580-6440

CHINA (Liaison Offices)

BEIJING
Beijing No. 5
14 Wu Lu Tong Road
De Sheng Men Wai
Tel.: (86-10) 202 4378
Telefax: (86-10) 202 4378

SHANGAI 200233
108-10 Astronartics Building
222 Cao Xi Road
Tel.: (86-21) 6472-5415
Telefax: (86-21) 6472-6814

SHENZHEN 518001
52, Tao Hua Road
Futian Free Trade Zone
SHENZHEN 518048
Tel.: (86-755) 359-0950
Telefax: (86-755) 359-1155

WANCHAI
22nd Floor - Hopewell centre
183 Queen's Road East,
Tel.: (852) 2861 5700
Telefax: (852) 2865 5000

INDIA (Liaison Offices)

BANGALORE 560052
Diners Business Service
26 Cunningham Road
Tel.: (91-80) 267 272
Telefax: (91-80) 261 133

NOIDA 201301
Liaison Office
Plot N. 2 & 3, Sector 16A
Institutional Area
Distt Ghaziabad UP
Tel.: (91-11) 853 0965/8
Telefax: (91-11) 855 6957

MALAYSIA

46000 PETALING JAYA, SELANGOR
Level 12-Std.D
PJ Tower (A)
AMCORP Trade Centre
No 18, Jalan Persiaran Barat
Off Jalan Timur
Tel.:(60-3) 758 1189
Telefax: (60-3) 758 1179

PENANG 10400
Unit 9-A, Lower Level 5
Hotel Equatorial
1 Jalan Bukit Jambul
Tel.: (60-4) 642 8291
Telefax (60-4) 642 8284

KOREA

SEOUL
19th Fl Kang Nam Building
1321-1 Seocho-dong Seocho-ku
Seoul
Tel.: (82-2) 3489-0114
Telefax:(82-2) 588-9030

TAE-GU 701-023
18th Floor Youngman Tower
111 Shinchun-3 Dong, Dong-Ku
Tel.: (82-53) 756-9583
Telefax:(82-53) 756-4463

SINGAPORE

SINGAPORE 2056
28 Ang Mo Kio - Industrial Park 2
Tel.: (65) 482 1411
Telefax: (65) 482 0240

TAIWAN

TAIPEI 106
20th Floor, #207
Tun Hua South Road, Section 2
Tel.: (886-2) 2378-8088
Telefax: (886-2) 2378-9188

THAILAND

BANGKOK 10110
Unit # 1315
54 Asoke Road
Sukhumvit 21
Tel. : (66-2) 260 7870
Telefax : (66-2) 260 7871

JAPAN

TOKYO 108
5 F Nisseki - Takanawa Blg. 2-18-10
Takanawa Minato-Ku
Tel.: (81-3) 3280-4120
Telefax: (81-3) 3280-4131

OSAKA 532
Shin-Osaka Second Mori Bldg..
3-5-36 Miyahara Yodogawa-Ku
Tel. (81-6) 397-4130
Telefax: (81-6) 397-4131

Information furnished is believed to be accurate and reliable. However, STMicroelectronics assumes no responsibility for the consequences of use of such information nor for any infringement of patents or other rights of third parties which may result from its use. No license is granted by implication or otherwise under any patent or patent rights of STMicroelectronics. Specifications mentioned in this publication are subject to change without notice. This publication supersedes and replaces all information previously supplied. STMicroelectronics products are not authorized for use as critical components in life support devices or systems without the express written approval of STMicroelectronics.

The ST logo is a trademark of STMicroelectronics

Intel® is a U.S. registered trademark of Intel Corporation.

Microsoft®, Windows® and Windows NT® are U.S. registered trademarks of Microsoft Corporation.

©1999 STMicroelectronics - All Rights Reserved.

Printed in France

Purchase of I^2C Components by STMicroelectronics conveys a license under the Philips I^2C Patent. Rights to use these components in an I^2C system is granted provided that the system conforms to the I^2C Standard Specification as defined by Philips.

STMicroelectronics Group of Companies
Australia - Brazil - Canada - China - France - Germany - Italy - Japan - Korea - Malaysia - Malta - Morocco - The Netherlands - Singapore - Spain - Sweden - Switzerland - Taiwan - Thailand - United Kingdom - U.S.A.

Recycled and chlorine free paper